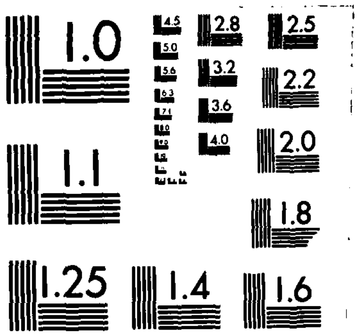
AD-A156 003 THE WEAPONS SUPPORT SYSTEM USER'S GUIDE(U) ANALYTIC SCIENCES CORP READING MA A R LESCHACK ET AL. 01 JUL 81 TASC-TR-1946-1 AFGL-TR-81-0225 F19628-80-C-0078 1/3

UNCLASSIFIED F/G 8/5 NL



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

AFGL-TR-81-0225

(2)

THE WEAPONS SUPPORT SYSTEM USER'S GUIDE

A. Richard LeSchack
Michael R. Tang

The Analytic Sciences Corporation
One Jacob Way
Reading, Massachusetts 01867

1 July 1981

Scientific Report No. 1

Approved for public release; distribution unlimited

Prepared For:

AIR FORCE GEOPHYSICS LABORATORY
AIR FORCE SYSTEMS COMMAND
United States Air Force
Hanscom AFB, Massachusetts 01731

CONTRACTOR REPORTS

This technical report has been reviewed and is approved for publication.

THEODORE E. WIRTANEN
Contract Manager

THOMAS P. ROONEY
Chief, Geodesy & Gravity Branch

FOR THE COMMANDER

DONALD H. ECKHARDT
Director
Earth Sciences Division

This report has been reviewed by the ESD Public Affairs Office (PA) and is releasable to the National Technical Information Service (NTIS).

Qualified requesters may obtain additional copies from the Defense Technical Information Center. All others should apply to the National Technical Information Service.

If your address has changed, or if you wish to be removed from the mailing list, or if the addressee is no longer employed by your organization, please notify AFGL/DAA, Hanscom AFB, MA 01731. This will assist us in maintaining a current mailing list.

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE

READ INSTRUCTIONS BEFORE COMPLETING FORM

1. REPORT NUMBER AFGL-TR-81-0225	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) THE WEAPONS SUPPORT SYSTEM USER'S GUIDE		5. TYPE OF REPORT & PERIOD COVERED Scientific Report No. 1
		6. PERFORMING ORG. REPORT NUMBER TR-1946-1
7. AUTHOR(s) A. Richard LeSchack Michael R. Tang		8. CONTRACT OR GRANT NUMBER(s) F19628-80-C-0078
9. PERFORMING ORGANIZATION NAME AND ADDRESS The Analytic Sciences Corporation One Jacob Way Reading, Massachusetts 01867		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS 320432AD
11. CONTROLLING OFFICE NAME AND ADDRESS Air Force Geophysics Laboratory Hanscom AFB, Massachusetts 01731 Monitor/Capt. Brian C. Mertz/LWG		12. REPORT DATE July 1981
		13. NUMBER OF PAGES 295
14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office)		15. SECURITY CLASS. (of this report) Unclassified
		15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public release; distribution unlimited.

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

Gravity
Gravity Data Evaluation
Weapons Support System
Data Smoothing
Spectrum Analysis
Multisensor Survey Simulation
GEOFAST

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

The Weapons Support System (WSS) was created as a research and development tool in the areas of gravity data evaluation, the generation of gravity products, and the study and analysis of geodetic and geophysical (G&G) effects on weapons system. The WSS will facilitate the design, development, and testing of methods, procedures, and algorithms, especially those that involve a user's online interaction, in real time, with a data base, computational facilities, and graphics devices. The WSS consists of:

DD FORM 1 JAN 73 1473 EDITION OF 1 NOV 65 IS OBSOLETE

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

BLOCK 20 (Contd)

- <u>HARDWARE</u>: - a carefully integrated combination of powerful minicomputer with associated peripheral devices, including user terminals and a color graphics display unit.

- <u>SYSTEMS SOFTWARE</u>: - a time-sharing virtual memory operating system to support a multiuser environment.

- <u>APPLICATIONS SOFTWARE</u>: - custom programs developed for the WSS, in these four application areas:

Evaluation of gravity data,
Data smoothing and analysis,
Multisensor survey simulation,
Rapid gravity field estimation (GEOFAST).

Accession For	
NTIS GRA&I	X
DTIC TAB	☐
Unannounced	☐
Justification	
By	
Distribution/	
Availability Codes	
Dist	Avail and/or Special
A-1	

TABLE OF CONTENTS

TABLE OF CONTENTS (Continued)

TABLE OF CONTENTS (Continued)

LIST OF FIGURES

LIST OF TABLES

1. THE WEAPONS SUPPORT SYSTEM USER'S GUIDE

1.1 INTRODUCTION

The Weapons Support System (WSS) was created as a research and development tool in the areas of gravity data evaluation, the generation of gravity products, and the study and analysis of geodetic and geophysical (G&G) effects on weapons systems. The WSS will facilitate the design, development, and testing of methods, procedures, and algorithms, especially those that involve a user's online interaction, in real time, with a data base, computational facilities, and graphics devices. At delivery time, the WSS consists of:

- HARDWARE:- a carefully integrated combination of powerful minicomputer with associated peripheral devices, including user terminals and a color graphics display unit

- SYSTEMS SOFTWARE:- a time-sharing virtual memory operating system to support a multiuser environment

- APPLICATIONS SOFTWARE:- custom programs developed for the WSS, in these four application areas:

  Evaluation of gravity data
  Data smoothing and analysis
  Multisensor survey simulation
  Rapid gravity field estimation (GEOFAST)

This introductory section gives an overview of the WSS hardware, the systems software, and the four major areas of application software, from the point of view of the user of

the WSS. Detailed operating instructions for each of the software areas follow the introduction.

1.2 THE WSS HARDWARE

The WSS hardware is based on a Digital Equipment Corporation (DEC) VAX-11/780 computer with 1.5 megabytes of high-speed storage. Additional memory is provided by two disk storage units:

- The DEC RM03 disk unit with a capacity of 67 megabytes
- The System Industries 9466 disk unit with a capacity of 300 megabytes.

The magnetic tape unit is a DEC TE16, which operates at a speed of 45 inches per second. A Printronix P600 line printer, with a capacity of 600 lines per minute, also serves as a medium-resolution plotter. The system uses a DECwriter III (LA 120) as the operator's console. The architecture is shown in Fig. 1.2-1.

Most user interaction will be with the five terminal devices currently supported by the WSS. Four of these are video data terminals, consisting of a keyboard and a DEC VT100 video display unit. Users enter data and commands, create and run programs, and view files and program output at these terminals. The fifth device is the Lexidata Display Unit, a color graphics monitor which can display maps, contour plots, and many other kinds of graphical output. Associated with the Lexidata unit is a trackball control device, to facilitate user control of and interaction with the displayed data. Any output displayed on the screen of the Lexidata unit can be

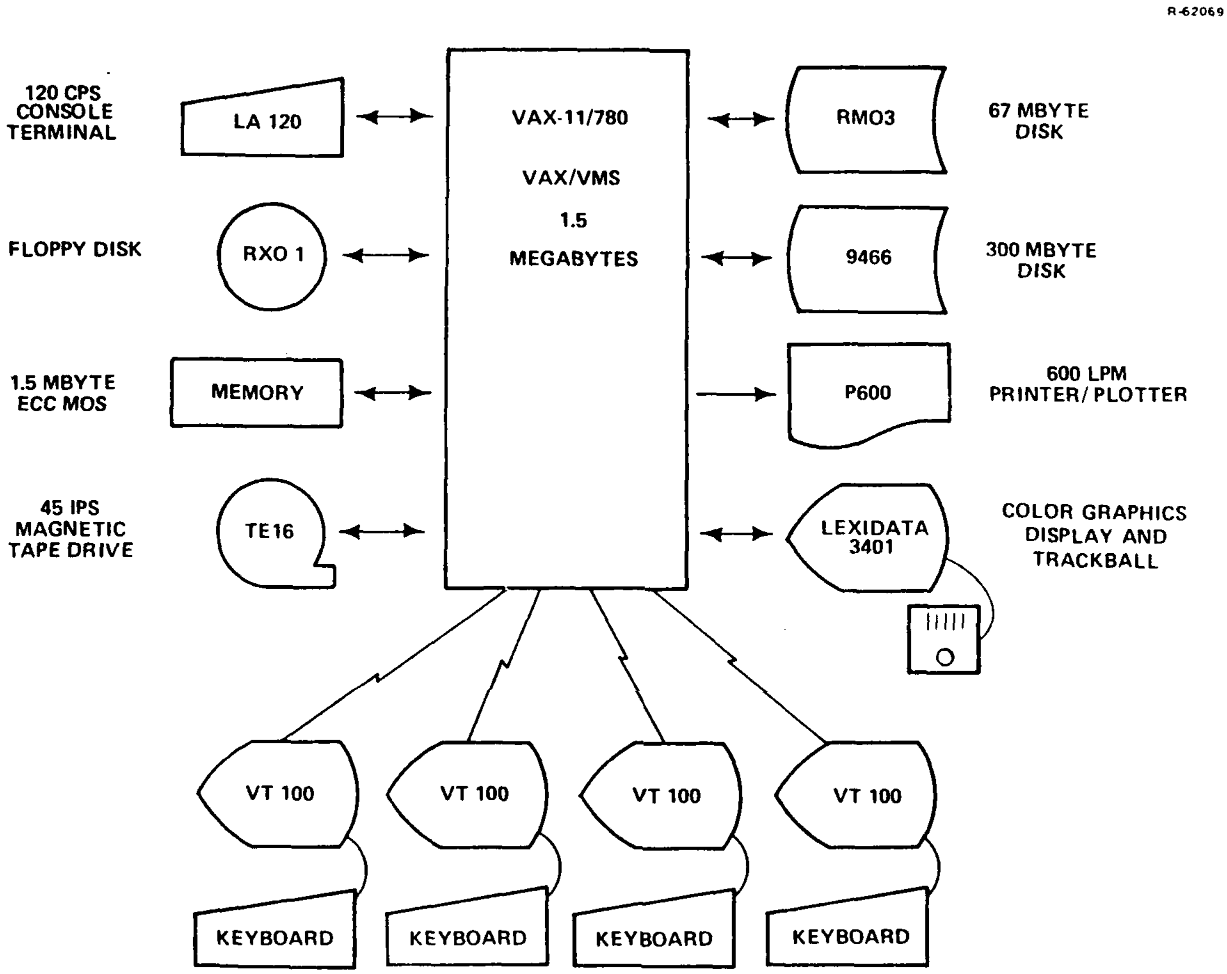


Figure 1.2-1 Weapons Support System Architecture

printed (although in black and white only) on the P600 line printer as a permanent hard copy.

1.3 THE WSS SYSTEMS SOFTWARE

The VAX-11/780 computer and its associated peripheral devices operate under the control of the VAX/VMS Operating

2 - Typing a 2 will invoke the cursor on the graphics terminal and allow the user to select individual points. To select a point, the user moves the cursor with the trackball over a point, and toggles the blue switch. The selected point will be marked in red. If there are several points near the cursor, the program will select the point nearest to the center of the cursor. The record in the gravity file corresponding to the selected point will be displayed at the VT100 terminal. The user will then have the option of saving the point in a separate file. Additional points can then be identified by moving the cursor and toggling the blue switch. To terminate this option, the user should toggle switch A (i.e., the left-most white switch).

3 - Typing a 3 will allow the user to select all points within a rectangular region on the scatter plot. Using the trackball, the user moves the cursor to one corner of the desired region and toggles the blue switch. A red "+" will appear on the graphics terminal. The user then moves the cursor to the diagonally opposite corner of the region, and again toggles the blue switch. Another "+" will appear at the point, and the rectangular region will be drawn using the two marked points. The points within the region will be marked in red, and a listing of the records of all the selected points will be displayed on the VT100 screen. The user will have the option of saving these points in a separate file. More regions can then be drawn by repeating the same steps. To terminate this option, the user should toggle switch A (i.e., the left-most white switch).

4 - Typing a 4 will allow the user to select entire tracks of data. This feature is most useful when analyzing ocean data, where the data tend to lie along tracks. As in selecting points within a rectangular region, the user employs the cursor to The selected points will be marked in red, and they will also be listed at the

Quit - Typing Quit (or Q) will cause the program to terminate.

If the user selects the zoom, scroll, and point selection option, the VT100 terminal screen will clear, and a map legend, which lists the source numbers and corresponding letters of the alphabet, will appear. The following menu will then be displayed on the VT100 screen:

ENTER 1 - ZOOM AND SCROLL
2 - USE CURSOR TO SELECT INDIVIDUAL POINTS
3 - USE CURSOR TO SELECT POINTS WITHIN A RECTANGULAR REGION
4 - USE CURSOR TO SELECT ENTIRE TRACKS
5 - USE CURSOR TO IDENTIFY INTERSECTION POINTS
99 - TO EXIT FROM THE PROGRAM

A description of each of the options will now be presented.

1 - By typing a 1, the user invokes the zoom and scroll option. By rolling the trackball, the user can move the scatter plot around the screen. To invoke the zoom feature, the user should move the area to be enlarged to the upper left corner of the graphics screen, and toggle the blue switch on the trackball unit. The user will then be prompted for the zoom factor at the VT100 terminal. The zoom factor is any integer value between 1 and 16. After the user enters the zoom level, the plot will be enlarged at the specified level. The program is still in the scroll mode, so that the picture can be moved about the screen. The blue switch can be toggled again, to enter a different zoom level. To freeze the screen and terminate the zoom and scroll option, the user should toggle switch A (i.e., the left-most white switch) on the trackball unit.

After reading the gravity file, the program will prompt the user for several inputs regarding latitude and longitude limits for the plot, plot title, and whether the user wants grid lines on the plot. All of the prompts have default values, which the user can select by pressing only the RETURN key.

The scatter plot is drawn using letters of the alphabet to represent data points from different sources. The major political boundaries and coastlines which lie within the limits of the scatter plot are drawn in red, and the latitude and longitude grid lines are drawn in blue.

When the scatter plot is completed, the following menu will appear on the VT100 screen:

```
TYPE <CR> - TO ZOOM, SCROLL AND IDENTIFY
            POINTS WITH THE CURSOR
   Replot - TO CREATE NEW PLOT
    Print - TO GET A HARD COPY OF PLOT
     Quit - TO HALT PROGRAM EXECUTION
```

When an option is selected, only the first letter of the user's response is examined. Hence, typing any character string that starts with P will produce a hard copy of the plot. A description of each option will now be given.

<CR> - Typing the RETURN key will allow the user to zoom, scroll, and select points using the cursor. Further details will be presented below.

Replot - By typing Replot (or R), the user can create a new plot on the graphics terminal. The user will be prompted for new latitude and longitude limits, and a new title.

Print - Typing Print (or P) will cause the scatter plot on the graphics terminal to be printed on the line printer. The user will then be asked to select again from this same menu.

Since the program uses the letters of the alphabet to distinguish different sources, a maximum of 26 unique sources can be identified within a given region.

2.3.3 Running the Scatter Plot Program

To run the scatter plot program, the user types the following command:

```
RUN SCATTER
```

The user will then be prompted to enter the name of the file containing the gravity survey data to be examined. The file name must include all qualifiers to identify the file uniquely. A complete file specification has the following format:

```
device: [directory]filename.type;version
```

The punctuation marks (colons, brackets, periods) are required to separate the various components of the file specification. If the input file is on the same directory as the user, only the filename and type need to be specified. The default file type is DAT, and if no version number is given, the file with the highest version number is used. Volume 2A of the VAX/VMS Reference Manuals contains a complete description of file specification.

If the word QUIT is typed for the input file name, the scatter plot program will stop execution and return control to VMS. The user should note that if the word QUIT is typed for many of the input prompts, the program will terminate and return control to VMS.

Before using any of the programs in the gravity data evaluation section, the user should make sure that the gravity file to be examined is in the WSS online format.

2.3 THE SCATTER PLOT PROGRAM

2.3.1 Purpose and Scope

The purpose of the scatter plot program is to display on the Lexidata graphics terminal the geographic locations of the gravity stations in a file containing gravity survey data. The scatter plot is superimposed on a map showing major political boundaries and coastlines. The map is drawn using a Mercator map projection. Both land and ocean survey data can be displayed using this program, and multiple sources within the same region are distinguished using different letters of the alphabet. The program is also set up to invoke the cursor, which can be used to identify individual points or subsets of points. The scatter plot program is run interactively from one of the VT100 terminals, and it requires the user to respond to several prompts and menu selection options.

2.3.2 Program Limitations

Since the scatter plot program requires the use of the Lexidata graphics terminal, the user should make sure that the graphics terminal is available before running the program. The user should also make sure that all the toggle switches on the trackball unit are in the OFF position.

The scatter plot program handles files with up to 5000 points. Larger files should be separated into smaller subfiles before processing with the scatter plot program.

convert from the card format to the WSS online format. This utility program, called REFORMAT, is included as part of the gravity data evaluation software. The program will first read the data stored in the 80 column card format. It will then allow the user to select subsets of the data based on:

- Geographic latitude/longitude boundaries
- Source identification number
- Any combination of the two.

The utility program will then create a data file containing the chosen subset of points in the WSS online format.

All of the programs which are part of the gravity data evaluation software use as input any file which is in the WSS online format. The WSS online format consists of the following data fields:

- Source number
- Latitude (decimal degrees)
- Longitude (decimal degrees)
- Elevation (meters)
- Observed gravity (less 976,000 mgal)
- Free-air anomaly (mgal)
- Bouguer anomaly (mgal).

The file, in WSS online format is read by the various programs using a formatted READ statement. The format for the READ statement is given by:

```
FORMAT(I5,2F10.4,F8.1,F8.2,2F7.1)
```

- The scatter plot program
- File editing and manipulation utilities
- The contour plot program
- The three-dimensional surface plot program
- Robust estimation and statistical plot program
- Ocean track-crossing adjustment programs

The Gravity Data Evalution User's Guide is organized in the following manner. The first section will describe the formatting requirements for the data files which will be processed by the gravity data evaluation programs. The next six sections will present detailed descriptions of how to implement the major sections of the gravity data evaluation software. The final section lists the error and warning messages which may occur during execution of the evaluation programs.

2.2 PREPROCESSING REQUIREMENTS

The Weapons Support System uses two different data formats for storing and processing gravity survey data. First, the WSS can read and store data from the DoD Gravity Library in the standard 80 column card format. For ease of use, however, a subset of the 80 column card format is commonly used as input to the gravity data evaluation software. This data format is called the WSS online format.

One of the implications of using the two different data formats, is that even if the 80 column card format should change, the WSS online format need not. However, the use of two data formats means that a utility program must be used to

2. USER'S GUIDE FOR THE GRAVITY DATA EVALUATION SOFTWARE

2 OVERVIEW OF THE GRAVITY DATA EVALUATION CAPABILITIES

The gravity data evaluation software is a collection of tools designed to assist the evaluator in the various tasks involved in analyzing gravity survey data. These tasks include:

- Elimination of erroneous and redundant data
- Verification of gravity base station
- Adjustment of systematic inconsistencies between sources
- *Identification and correction of recoverable errors*
- Assignment of accuracy measures to each source.

The various programs which make up the gravity data evaluation software make extensive use of the Lexidata graphics terminal. The graphical displays are extremely useful in that they provide the evaluator with a quick-look capability for making rapid decisions concerning the data. However, these programs also rely heavily on the evaluator's judgment and experience.

The gravity data evaluation software is divided into major sections. They are:

capabilities of the multisensor survey simulation program

- Running The MULTISENS program -- showing the user how to run the multisensor survey simulation program

- Running The MULTIJOB Program -- showing the user how to create the job stream required to run MULTISENS

- Running the MULTIPLOT Program -- showing the user how to generate plots of MULTISENS results.

The Geofast Estimation Software section of the WSS User's Guide is organized as follows:

- Overview -- describing, in general terms, the capabilities and features of the GEOFAST package

- *Program Execution -- showing the user* how to run the interactive front end program (GEOFAST) that prompts for required input parameters and sets up the batch jobs to do GEOFAST estimation, and explaining, in summary form, the operation of the two components of the estimation system: - GEOCOV3, which does preprocessing of covariances and carries out the transformation to the frequency domain; and GEOEST3, which uses the output files generated by GEOCOV3 to compute the required estimates

- GEOPLOT -- describing the use of the interactive program (GEOPLOT), which displays the results of GEOFAST runs.

the various operations involved in the selection of tracks, determination of gravity values at the intersection points, computation of track adjustments, and application of the computed adjustments to the gravity data file.

The section concludes with a summary of possible error messages that the user may encounter.

The Spectrum Analysis and Smoothing Software section of the WSS User's Guide is organized as follows:

- Introduction -- describing, in general terms, the four programs that are available for data smoothing and spectrum analysis

- Data Formats -- explaining the standard WSS time series data file conventions

- Data Selection -- showing the user how to run the program that examines data files, selects subsets, smooths, and resamples

- AR Modeling -- showing the user how to run the program that carries out autoregressive modeling and plots power spectra and coherency

- Periodogram Spectral Analysis And Plotting -- showing the user how to run the program (FFT) that carries out periodogram analysis on the master data file or a selected subset, creating an averaged periodogram with associated error statistics; and the program (PLOTFFT) to generate periodogram plots.

The Multisensor Survey Simulation Software section of the WSS User's Guide is organized as follows:

- Applications Of The Multisensor Survey Simulation Software -- describing, in general terms, the various options and

control to the VAX/VMS operating system by entering QUIT as a response to most interactive prompts.

The User's Guide is intended to be a system introduction and a working guide for use during terminal sessions. A brief overview will now be given for each of the WSS applications software functions.

The Gravity Data Evaluation Software section of the WSS User's Guide is organized as follows:

- Overview -- describing the gravity data evaluation software in general terms
- Preprocessing Requirements -- explaining the standard data format used for the point gravity files
- Scatter Plotting -- showing the user how to run the program (SCATTER) that implements the scatter plotting features, including zoom, scroll, and the use of the cursor to select individual points and points within a rectangular area
- File Editing and Manipulation -- showing how to use the features of the VAX/VMS system editor (EDT) to accomplish file editing, using the common station compare as an example
- Contour Plotting -- showing the user how to run the program (CONPLOT) that generates a labeled contour plot on the Lexidata Graphics Terminal
- Three-Dimensional Plotting -- showing the user how to run the program (SURPLOT) that generates labeled three-dimensional plots
- Robust Estimation and Statistical Plotting -- showing the user how to run the program (STATPLOT) that implements the various robust estimators and generates statistical plots like the QQ Plot
- Track Crossing Adjustment Tools -- showing the user how to run the four programs that implement

The TASC Graphics Software Package originated with the National Center for Atmospheric Research and has been extensively modified by TASC for use with the WSS. It provides the following features:

- Automatic generation of various kinds of graphs
- Contour plotting
- Three-dimensional plotting
- Map generation
- Generation of halftone pictures
- Velocity displays
- Generation of characters and printed titles.

The TASC Graphic Software Package is documented in Ref. 7.

1.4 THE WSS APPLICATIONS SOFTWARE

The WSS Applications Software includes the following major capabilities, each of which is documented in detail in the sections to follow:

- Gravity Data Evaluation
- Spectrum Analysis and Smoothing Software
- Multisensor Survey Simulation Software
- GEOFAST Estimation Software.

During execution of WSS application software, the user may stop execution of the application software and return

These are used extensively by the WSS applications programs to be described in this User's Guide. Th' are also available to WSS users who will be writing their own programs.

IMSL, a carefully tested and documented collection of 495 FORTRAN programs, includes the following categories:

- Analysis of Variance
- Basic Statistics
- Categorized Data Analysis
- Differential Equations and Quadrature
- Eigensystem Analysis
- Forecasting, Time Series, and Transforms
- Random Numbers
- Interpolation, Approximation, and Smoothing
- Linear Algebraic Equations
- Mathematical and Statistical Special Functions
- Nonparametric Statistics
- Observation Structure and Multivariate Analysis
- Regression Analysis
- Sampling
- Utility Functions
- Vector and Matrix Arithmetic
- Zeros, Extrema, and Linear Programming.

The IMSL programs are described in a three-volume reference manual published by International Mathematical and Statistical Libraries, Inc., delivered with the system.

System, Version 2.2, which is designed to permit the simultaneous use of the computer and associated hardware by many users. The operating system is described in summary form, at the level required by the WSS user, in a Digital Equipment Corporation document entitled VAX/VMS Primer, copies of which will be delivered with the WSS. The WSS user will find basic instructions in the Primer for logon and logoff procedures, and for using the VT100 terminal to communicate with the system. Beyond these basic concepts, most WSS users will require knowledge of only a few features of the operating system in order to use the existing WSS capabilities, since detailed and self-contained directions are provided for each applications program in this User's Guide. The most important of these features for the WSS user is the EDITOR, a powerful and easy-to-use tool for the creation, examination, and modification of files -- a term that includes data as well as programs. The use of EDIT commands to manipulate gravity data files in the context of gravity data evaluation is described below in section 2.2.4 (File Editing and Manipulation).

Detailed documentation of the VAX/VMS Operating System is available in the form of a multivolume set of manuals. The Primer, as well as the Information Directory and Index in Volume One of the VAX/VMS Documentation, will direct the user to the appropriate source of detailed information about any aspect of the Operating System.

Also included under the heading of systems software are the following software packages:

- The International Mathematical and Statistical Library (IMSL)
- The TASC Graphics Software Package.

VT100 terminal. The user will have the option of saving them. To terminate this option, the user should toggle switch A (i.e., the left-most white switch).

5 - Typing a 5 will allow the user to determine the coordinates of track intersection points. Again, this is most useful when analyzing ocean data. The user should move the cursor over the intersection point and toggle the blue switch. The latitude and longitude coordinates of the intersection point will appear on the VT100 screen. Further intersection points can then be determined. To terminate this left-most white switch).

99 - Typing a 99 will cause the program to terminate.

Note that for each of the user-selected options, the user terminates an option by toggling switch A. The user menu selection options will then appear on the screen. Before selecting another option, the user should make sure that switch A is in the OFF position.

If an invalid switch is toggled during execution of any of the user selected options, the following message will appear on the screen:

```
***** USER ERROR *****
AN INVALID SWITCH HAS BEEN TOGGLED
RESET TOGGLE SWITCH AND TRY AGAIN
```

The user should move the invalid switch to the OFF position before toggling the correct switch.

2.4. FILE EDITING AND MANIPULATION UTILITIES

2.4.1 Introduction

The VAX/VMS operating system has an extensive set of commands for editing and manipulating data files and user programs. These commands are part of the Digital Command Language (DCL). The DCL command language contains commands for performing such operations as:

- Changing and/or modifying files
- Printing files on the line printer
- Deleting one or more files
- Copying sections of one file into another file
- Sorting files on specified fields.

The various DCL commands are fully documented in Volume 2A of the VAX/VMS Reference Manuals. The documentation is also available at the VT100 terminal by use of the HELP command. To invoke the HELP command, the user simply types the word HELP, followed by the command for which more information is required. The HELP command will respond with a summary of the format of the particular command or a list of the command's valid qualifiers. For example, to obtain more information about the COPY command, the user should type:

```
HELP COPY
```

The system responds by displaying at the terminal a summary of the COPY command and keywords to enter as parameters to the

HELP command to obtain additional information. As an example, if the user types:

HELP COPY/EXTENSION

the system will respond with information concerning the extensions to be added to the output file. If the user enters:

HELP COPY QUALIFIERS

the HELP command will display a description of each of the COPY command qualifiers.

Two of the file manipulation utilities which are used very frequently are the interactive text editor and the sorting facility. A description of these two utilities will now be presented.

2.4.2 Interactive Text Editor

The interactive text editor allows the user to examine, create, and/or modify data files or user programs from the VT100 terminal. The VAX/VMS operating system has two interactive text editors. They are called SOS and EDT. The SOS editor is a line editor, while the EDT editor is a full-screen editor. The EDT editor is the more useful of the two. Complete documentation for both of these editors is given in Volume 3A of the VAX/VMS Reference Manuals.

The easiest way to learn how to use the EDT editor is to run the EDT Editor Computer Aided Instruction minicourse, which is called EDTCAI. This minicourse is presented interactively at the computer terminal. The computer will display

information about the EDT editor on the VT100 terminal screen, allow the user to practice entering EDT commands, and ask the user questions about what has been learned. For more information about running EDTCAI, the user should talk to the systems manager.

2.4.3 SORT/MERGE Facility

The SORT/MERGE facility allows users of the VAX/VMS operating system to reorder data files in either ascending or descending order. The user can sort several files using the same field, and then merge the files into one sorted file. The user can run SORT/MERGE interactively from the terminal, as a batch job, or as part of a user program.

The purpose of this section is to present a brief introduction to some of the useful capabilities of the SORT/MERGE facilities. The user should refer to Volume 3A of the VAX/VMS Reference Manuals for a detailed description of the various features in the SORT/MERGE program.

The SORT command has the following general form:

```
SORT/KEY=([qualifiers]) input-file(s) output-file
```

The qualifiers specify which data field is to be used as the sort key. The user can specify up to 10 different input files, which must exist either on the disk or magnetic tape. The output file contains the sorted records.

For example, assume that the user has a WSS gravity input file called SRCE4242.DAT. (See Section 2.2 for a description of the WSS online file format). To sort this file on the elevation field, the user types:

```
SORT/KEY=(POSITION=26,SIZE=8) SRCE4242.DAT SORT.DAT
```

The output file, SORT.DAT, will contain all the records of the file SRCE4242.DAT sorted by elevation in ascending order.

If more than one input file is specified, the input files will all be sorted on the specified field, and then merged into a single sorted file. This could be used to facilitate a Common Station Compare. For example, if the user has two files of gravity survey data covering the same region, FILE1.DAT and FILE2.DAT, then the following command:

```
SORT/KEY=(POS=6,SIZE=10)/KEY=(POS=16,SIZE=10) -
     FILE1.DAT,FILE2.DAT  COMMON.DAT
```

will first sort the two input files by latitude and longitude, and then merge the two sorted files into COMMON.DAT. Hence, pairs of points which are geographically close to each other will appear as sequential records in the output file.

Similarly, if several data files have already been sorted on the same field, they can be merged into a single sorted file using the MERGE command. The user should look in the reference manual for details on the MERGE command.

2.5 THE CONTOUR PLOT PROGRAM

2.5.1 Purpose and Scope

The purpose of the contour plot program is to display on the Lexidata graphics terminal a contour plot of a specified data field from a file containing gravity survey data. The program is set up to overlay the contour plot with a scatter

plot, which shows the geographic locations of the individual gravity stations of the gravity survey file. Both land and ocean survey data can be displayed using this program, and the data need not be evenly spaced. The program is also set up to invoke the cursor to identify individual points or subsets of points. The contour plot program is run interactively from the one of the VT100 terminals, and it requires the user to respond to several prompts and menu selection options.

2.5.2 Program Limitations

Since the contour plot program requires the use of the Lexidata graphics terminal, the user should make sure that the graphics terminal is available before running the program. Also, the user should make sure that all the toggle switches located on the trackball unit are in the OFF position.

The contour plot program handles files with up to 5000 points. Larger files should be separated into smaller subfiles before processing with the contour plot program. The contour plot program also requires a minimum of two data points. However, to ensure that the contour plot adequately represents the data field, the file should contain at least 10 data points.

2.5.3 Running the Contour Plot Program

To run the contour plot program, the user types the following command:

```
RUN CONPLOT
```

The user will then be prompted to enter the file name containing the gravity survey data to be examined. The file name must include all qualifiers to identify the file uniquely. A complete file specification has the following format:

```
device:[directory]filename.type;version
```

The punctuation marks (colons, brackets, periods) are required to separate the various components of the file specification. If the input gravity file is located on the same directory as the user, only the filename and type need to be specified. The default type is DAT, and if no version number is specified, then the file with the highest version number is used. Volume 2A of the VAX/VMS documentation contains a complete description of file specification.

If the word QUIT is typed for the input file name, the contour plot program will stop execution and return control to VMS. The user should note that if the word QUIT is typed for many of the input prompts, the program will terminate and return control to VMS.

After reading the gravity file, the program will prompt the user for inputs regarding latitude and longitude limits for the contour plot. All of the prompts have default values, which the user can select by pressing the RETURN key.

The user will then receive the following message at the terminal:

```
WHAT TYPE OF CONTOUR PLOT?
TYPE E 1 ELEVATION PLOT
     F 2 FREE-AIR ANOMALY PLOT
     B 3 BOUGUER ANOMALY PLOT
```

The default value is a contour plot of the Bouguer anomaly field. The user can select the default by pressing the RETURN key.

The user is then requested to enter a title for the plot. The default title is the input file name, but the user may also wish to specify the field being contoured. That is, a more useful title may be something like SOURCE 4242 BOUGUER ANOMALY. The plot title must be less than 40 characters.

The following message will then appear on the screen:

```
DO YOU WISH TO SELECT YOUR OWN CONTOUR LEVELS ?
ENTER Yes OR No
DEFAULT - No : LET PROGRAM CHOOSE "NICE" CONTOUR LEVELS
```

If the user selects the default option, the program will select the contour levels to be drawn. The program will select between 10 and 30 evenly spaced contour levels which will cover the range of data values.

The user can specify the contour levels by typing "YES" in response to the above prompt. If this option is selected, the user will be prompted for a minimum contour level, a maximum contour level, and the increment between the contour levels. The user can specify the value 0.0 for the increment value. This will cause the program to choose between 10 and 30 contour levels which lie between the maximum and minimum.

The contour plot is drawn so that the major contour levels are in green and the minor contour levels are in red. Positive contour levels are drawn as solid lines, while negative contours are drawn as dashed lines. Local minimum and maximum are labeled on the contour plot with an L and H, respectively. The geographic locations of the actual data points are marked on the plot with blue "+"s.

When the contour plot is completed, the following menu will appear on the VT100 screen:

```
TYPE <CR> - TO ZOOM, SCROLL AND IDENTIFY
            POINTS WITH THE CURSOR
   Replot - TO CREATE NEW PLOT
    Print - TO GET A HARD COPY OF PLOT
     Quit - TO HALT PROGRAM EXECUTION
```

When an option is selected, only the first letter of the user's response is examined. Hence, typing any character string that starts with P will produce a hard copy of the plot. A description of each option will now be given:

<CR> - Typing the RETURN key will allow the user to zoom, scroll, and select points using the cursor. Further details will be presented next.

Replot - By typing Replot (or R), the user can create a new plot on the graphics terminal. The user will be prompted for new latitude and longitude limits, a different data field, a new title, and new contour levels. As before, default values are available for the various prompts.

Print - Typing Print (or P) will cause the contour plot on the graphics terminal to be printed on the line printer. The user will then be asked to select agai[n] from this same menu.

Quit - Typing Quit (or Q) will cause the program to terminate.

If the user selects the zoom, scroll and point selection option, the following menu will appear on the screen:

```
ENTER 1 - ZOOM AND SCROLL
      2 - USE CURSOR TO SELECT INDIVIDUAL POINTS
      3 - USE CURSOR TO SELECT POINTS WITHIN
              A RECTANGULAR REGION
      4 - NEW PLOT
     99 - TO EXIT FROM THE PROGRAM
```

A decription of each of the options will now be presented.

1 - By typing a 1, the user invokes the zoom and scroll option. By rolling the trackball, the user can move the contour plot around the screen. To invoke the zoom feature, the user should move the area to be enlarged to the upper left corner of the graphics screen, and toggle the blue switch on the trackball unit. The user will then be prompted for the zoom factor at the VT100 terminal. The zoom factor is any integer value between 1 and 16. After the user enters the zoom level, the plot will be enlarged at the specified level. The program is still in the scroll mode, so that the picture can be moved about the screen. The blue switch can be toggled again, to enter a different zoom level. To freeze the screen and terminate the zoom and scroll option, the user should toggle switch A (i.e., the left-most white switch) on the trackball unit.

2 - Typing a 2 will invoke the cursor on the graphics terminal and allow the user to select individual points. To select a point, the user moves the cursor with the trackball over a point, and toggles the blue switch. The selected point will be marked in red. If there are several points near the cursor, the of the cursor. The record in the gravity file corresponding to the selected point will be displayed at the VT100 terminal. The user will then have the option of saving the point in a separate file. Additional points can then be identified by moving the cursor and toggling the blue switch. To terminate this option, the user should toggle switch A (i.e., the left-most white switch).

3 - Typing a 3 will allow the user to select all points within a rectangular region on the contour plot. Using the trackball, the user moves the cursor to one corner of the desired region and toggles the blue switch. A red "+" will appear on

the graphics terminal. The user then moves the cursor to the diagonally opposite corner of the region, and again toggles the blue switch. Another "+" will appear at the point, and the rectangular region will be drawn using the two marked points. The points within the region will be marked in red, and a listing of the records of all the selected points will be displayed on the VT100 screen. The user will have the option of saving these points in a separate file. More regions can then be drawn by repeating the same steps. To terminate this option, the user should toggle switch A (i.e., the left-most white switch).

4 - By typing a 4, the user can create a new contour plot on the graphics terminal. The user will be prompted for new latitude and longitude limits, a different data field, a new title, and new contour levels. As before, default values are available for the various prompts.

99 - Typing a 99 will cause the program to terminate.

Note that for each of the user-selected options, the user terminates an option by toggling switch A. The user menu selection options will then appear on the screen. Before selecting another option, the user should make sure that switch A is in the OFF position.

If an invalid switch is toggled during execution of any of the user selected options, the following message will appear on the screen:

```
***** USER ERROR *****
AN INVALID SWITCH HAS BEEN TOGGLED
RESET TOGGLE SWITCH AND TRY AGAIN
```

The user should move the invalid switch to the OFF position before toggling the correct switch.

2.6. THE THREE-DIMENSIONAL SURFACE PLOT PROGRAM

2.6.1 Purpose and Scope

The purpose of the three-dimensional surface plot program is to display on the Lexidata graphics terminal a surface plot of a specified data field from a file containing gravity survey data. Both land and ocean survey data can be displayed using this program, and the data need not be evenly spaced. The three-dimensional surface plot program is run interactively from the one of the VT100 terminals, and it requires the user to respond to several prompts and menu selection options.

2.6.2 Program Limitations

Since the surface plot program requires the use of the Lexidata graphics terminal, the user should make sure that the graphics terminal is available before running the program. The user should also make sure that all the toggle switches located on the trackball unit are in the OFF position.

The surface plot program handles files with up to 5000 points. Larger files should be separates into smaller subfiles before processing with the surface plot program. The surface plot program also requires a minimum of two data points. However, to ensure that the surface plot adequately represents the data field, the file should contain at least 10 data points.

2.6.3 Running the Three-Dimensional Surface Plot Program

To run the surface plot program, the user types the following command:

```
RUN SURPLOT
```

The user will then be prompted to enter the file name containing the gravity survey data that are to be examined. The file name must include all qualifiers to identify the file uniquely. A complete file specification has the following format:

```
device:[directory]filename.type;version
```

The punctuation marks (colons, brackets, periods) are required to separate the various components of the file specification. If the input gravity file is located on the same directory as the user, only the filename and type need to be specified. The default type is DAT, and if no version number is specified, the file with the highest version number is used. Volume 2A of the VAX/VMS Reference Manuals contains a complete description of file specification.

If the word QUIT is typed for the input file name, the surface plot program will stop execution and return control to VMS. The user should note that if the word QUIT is typed for many of the input prompts, the program will terminate and return control to VMS.

After reading the gravity file, the program will prompt the user for inputs regarding latitude and longitude limits for the surface plot. All of the prompts have default values, which the user can select by pressing the RETURN key.

The user will then receive the following message at the terminal:

```
WHAT TYPE OF SURFACE PLOT?
TYPE 1 - ELEVATION PLOT
     2 - FREE-AIR ANOMALY PLOT
     3 - BOUGUER ANOMALY PLOT
```

e default value is a surface plot of the Bouguer anomaly eld. The user can select the default by pressing the RETURN :y.

The user is then requested to enter a title for the lot. The default title is the input file name, but the user ay change the title to any character string (maximum length f 40 characters).

The following message will then appear on the VT100 creen:

```
ENTER THE TWO VIEWING ANGLES FOR THE LINE OF SIGHT
    ANGH : ANGLE (DEGREES) IN THE X-Y PLANE TO THE
           LINE OF SIGHT (COUNTERCLOCKWISE FROM THE
           POSITIVE X-AXIS)
    ANGV : ANGLE (DEGREES) FROM THE X-Y PLANE TO THE
           LINE OF SIGHT.  POSITIVE ANGLES ARE ABOVE
           MIDDLE Z; NEGATIVE, BELOW.
```

'or the surface plot program, the longitude scale is considered s the x-axis, and the latitude scale is the y-axis. The de-'ault values are 45 deg for ANGH and 15 deg for ANGV. The efault values can be selected by typing the RETURN key.

When the surface plot is completed, the following ienu will appear on the VT100 screen:

```
TYPE <CR> - TO ZOOM, SCROLL AND CHANGE
               VIEWING ANGLE
   Replot - TO CREATE NEW PLOT
    Print - TO GET A HARD COPY OF PLOT
     Quit - TO HALT PROGRAM EXECUTION
```

When an option is selected, only the first letter of he user's response is examined. Hence, typing any character string that starts with P will produce a hard copy of the plot. description of each option will now be given.

in each segment using the cursor, and write the points from each segment into the same file. To terminate the track selection option, the user should toggle switch A (i.e., the left-most white switch).

5 - Typing a 5 will allow the user to identify track intersection points. To identify an intersection point, the user should move the cursor over the intersection point and toggle the blue switch. The latitude and longitude coordinates of the point will appear on the VT100 screen. Other intersection points can then be identified. To terminate this option, the user should toggle switch A (i.e., the left-most white switch).

The other options of the scatter plot are also avail-ble to the user for the analysis of ocean data. The scatter lot user's guide (Sec. 2.3) describes in detail how to use he other options.

Program CLEANUP

The purpose of the program CLEANUP is to remove the xtra points which are sometimes selected as part of an ocean rack and to sort the track by latitude and longitude. To run he CLEANUP program, the user should type:

```
RUN CLEANUP
```

The user will then receive the following prompt at he terminal:

```
ENTER INPUT FILE NAME OR
     QUIT TO HALT PROGRAM EXECUTION
```

data evaluation software and has its own user's guide. The user should read the scatter plot user's guide (Sec. 2.3) before using program SCATTER with ocean data. Two of the features of the scatter plot program specifically designed for ocean data will now be discussed in detail.

When the scatter plot is completed and the user has selected the zoom, scroll, and point selection option, the following menu will appear on the VT100 screen:

```
ENTER 1 - ZOOM AND SCROLL
      2 - USE CURSOR TO SELECT INDIVIDUAL POINTS
      3 - USE CURSOR TO SELECT POINTS WITHIN
              A RECTANGULAR REGION
      4 - USE CURSOR TO SELECT ENTIRE TRACKS
      5 - USE CURSOR TO IDENTIFY INTERSECTION POINTS
     99 - TO EXIT FROM THE PROGRAM
```

Options 4 and 5 are specifically related to ocean data analysis, so a detailed description of these options will be presented here again.

4 - Typing a 4 will allow the user to select entire tracks of data. The user should use the cursor to mark the starting and stopping points of the track. The selected points will be marked in red, and the user will have the option to save them. Two problems can arise in selecting a track in this manner. First, because of the algorithm used to select the points along a track, the program may select points from other tracks at the intersection points. The user should still write all the selected points to a file. The extraneous points will be removed in another program. The second problem in selecting tracks by straight lines is that real data hardly ever occur along perfectly straight lines. One technique which avoids this problem is to divide the track into a few linear segments. Select the points

correspond exactly with the four major steps for solving the track-crossing adjustment problem. A description of each of these programs will be presented in Section 2.8.3. Section 2.8.4 will then describe how to combine these four programs in an organized manner to complete the four steps in solving the ocean track-crossing adjustment problem.

2.8.2 Program Limitations

The scatter plot program, which is one of the ocean track-crossing adjustment programs, requires the use of the Lexidata graphics terminal. The user should make sure that the graphics terminal is available before running the scatter plot program. The user should also make sure that all the toggle switches located on the trackball unit are in the OFF position.

The ocean track-crossing adjustment programs are able to handle files with up to 5000 points. Larger files should be separated into smaller subfiles before processing with the ocean track-crossing adjustment programs. The program which sets up and solves the linear program can process a maximum of 25 tracks and 50 intersection points.

2.8.3 Ocean Track-Crossing Adjustment Program Descriptions

This section will present a description of the four programs which make up the ocean track-crossing adjustment software.

Program SCATTER

Program SCATTER produces a scatter plot of a selected file of ocean gravity data. This program is part of the gravity

One of the evaluation tasks specifically associated with ocean data is the determination and, if possible, adjustment of the systematic discrepancies at crossings of tracks belonging to different sources. The track adjustment process typically begins with a track of known (or assumed) high accuracy, or a track tied reliably to ground data (dockside calibration). The adjustments then work outward in cantilever style from the track or tracks assumed correct. The process as it is carried out in practice has subjective elements and is known to lead to different results in the hands of different evaluators.

An alternative approach to the track adjustment problem is to consider all tracks and intersection points within a given region simultaneously, and determine track adjustment factors on a global scale. The track adjustment problem can then be formulated as a linear programming model. The solution to the linear program is a set of track adjustments which minimize the maximum absolute discrepancy at the intersection points, subject to any constraints which the user may impose on the individual track adjustments. A detailed description of this method is given in Refs. 5 and 6.

Solving the ocean track-crossing adjustment problem consists of four major steps, which can be summarized as follows:

- Selection of individual tracks
- Determination of track intersection points
- Determination of gravity anomaly at the intersection points
- Setting up and solving the linear programming model

To accomplish these four steps, the user must run four different programs. However, the four programs do not

FLATLABS - The least absolute value with flattened weights of the displayed data is computed and displayed on the VT100.

M-ESTIMATE - The M-estimate (type 1 or 2) is computed for the plotted points. The user will be prompted to enter the type, with the default being the last type used. The original default is type 1.

SINE-ESTIMATE - The sine estimate of the plotted points is computed and presented on the VT100.

After the user enters a RETURN (<CR>), the screen will clear, and the list of available commands will appear at the top of the screen, without any description. The number of points being analyzed by the robust estimator functions will also be displayed. The user can now enter any valid robust estimator command, and the results will be displayed on the VT100 screen. If there fewer than 3 data points, some of the estimators will not function properly. Warning messages will be displayed if such cases arise. Further information on the robust estimators is given in Refs. 6 and 6.

2.8 OCEAN TRACK-CROSSING ADJUSTMENT PROGRAMS

2.8.1 Purpose and Scope

The purpose of the ocean tracking-crossing adjustment programs is to provide the user with a set of tools to aid in the evaluation of ocean gravity data. The evaluation of ocean data presents specific problems not encountered with land data. Gravity measurements in ocean areas are generally made along intersecting tracks, with large in-between areas in which no data are available.

BICKEL - The Bickel-Hodges estimate of the plotted points is computed and displayed on the VT100.

MEDIAN - The median, extremes, and the upper and lower quartiles are displayed on the VT100.

HODGES - The Hodges-Lehmann estimate of the plotted points is computed and presented on the VT100. Since this estimation technique computes the median of all possible pairs of values, it is limited to working on sets of data with less than 1000 points. In addition, if the number of points is large, it may take a long time to compute the estimate.

WINSOR - A Winsorized mean of the displayed data is computed and presented on the VT100. The user will be prompted to enter the percentage, which must be between 0 and 50 percent. The value is entered as a percent (e.g., ten percent is entered as a 10). The default percentage is what was used the last time, with 10 percent as the first default.

TRIMMED - The trimmed mean of the plotted points is computed and displayed on the VT100. The user will be prompted to enter the percentage, which must lie between 0 and 50 percent. The value is entered as a percent (e.g., ten percent is entered as a 10). The default percentage is what was used the last time, with 10 percent as the first default.

ADAPTIVE - An adaptive trimmed mean of the plotted points is computed and presented on the VT100.

BIWEIGHT - The biweight estimate for the plotted points is computed and displayed on the VT100. The user will be prompted to enter the weighting factor, which must lie between 2 and 15. The default value is determined from the last use, with 5 being the first default.

Note that for each of the user-selected options, the user terminates an option by toggling switch A. The user menu selection options will then appear on the screen. Before selecting another option, the user should make sure that switch A is in the OFF position.

If an invalid switch is toggled during execution of any of the user selected options, the following message will appear on the screen:

```
***** USER ERROR *****
AN INVALID SWITCH HAS BEEN TOGGLED
RESET TOGGLE SWITCH AND TRY AGAIN
```

The user should move the invalid switch to the OFF position before toggling the correct switch.

If the user selects the robust estimator option, a list of valid commands for the robust estimators, along with a brief description of their meaning, will appear on the VT100 screen. To invoke any of the robust estimator commands, the user must enter at least the first three letters of the command name. The list includes the following commands:

DONE - The user will be returned to the user-selection options menu shown above

QUIT - Program execution is terminated.

HELP - A list of valid commands is displayed with a brief explanation of what they do. This is the same list presented on entry to the robust estimator section.

MEAN - The mean and standard deviation of the plotted points are computed and displayed on the VT100.

near the cursor, the program will select the point nearest to the center of the cursor. The record in the gravity file corresponding to the selected point will be displayed at the VT100 terminal. The user will then have the option of saving the point in a separate file. Additional points can then be identified by moving the cursor and toggling the blue switch. To terminate this option, the user should toggle switch A (i.e., the left-most white switch).

3 - Typing a 3 will allow the user to select all points within a rectangular region on the statistical plot. Using the trackball, the user moves the cursor to one corner of the desired region and toggles the blue switch. A red "+" will appear on the graphics terminal. The user then moves the cursor to the diagonally opposite corner of the region, and again toggles the blue switch. Another "+" will appear at the point, and the rectangular region will be drawn using the two marked points. The points within the region will be marked in red, and a listing of the records of all the selected points will be displayed on the VT100 screen. The user will have the option of saving these points in a separate file. More regions can then be drawn by repeating the same steps. To terminate this option, the user should toggle switch A (i.e., the left-most white switch).

4 - Entering a 4 will remove the selected points (those marked in red) and replot the rest of the data using the same type of plot (i.e., either a QQ plot or ECDF plot). The user will then be prompted with the first menu selection options.

5 - By typing a 5, the user can create a new plot on the graphics terminal. The user will be prompted for the type of plot and data field. All the data points from the input file will be used in constructing the new plot, not the subset of points that command 4 would plot.

99 - Typing a 99 will cause the program to terminate.

Quit - Typing Quit (or Q) will cause the program to terminate.

If the user selects the zoom, scroll and point selection option, the following menu will appear on the VT100 screen:

```
ENTER 1 - ZOOM AND SCROLL
      2 - USE CURSOR TO SELECT INDIVIDUAL POINTS
      3 - USE CURSOR TO SELECT POINTS WITHIN
            A RECTANGULAR REGION
      4 - REMOVE SELECTED POINTS AND REPLOT
      5 - NEW PLOT
     99 - TO EXIT FROM THE PROGRAM
```

A decription of each of the options will now be presented.

1 - By typing a 1, the user invokes the zoom and scroll option. By rolling the trackball, the user can move the statistical plot around the screen. To invoke the zoom feature, the user should move the area to be enlarged to the upper left corner of the graphics screen, and toggle the blue switch on the trackball. The user will then be prompted for the zoom factor at the VT100 terminal. The zoom factor is any integer value between 1 and 16. After the user enters the zoom level, the plot will be enlarged at the specified level. The program is still in the scroll mode, so that the picture can be moved about the screen. The blue switch can be toggled again, to enter a different zoom level. To freeze the screen and terminate the zoom and scroll option, the user should toggle switch A (i.e., the left-most white switch) on the trackball unit.

2 - Typing a 2 will invoke the cursor on the graphics terminal and allow the user to select individual points. To select a point, the user moves the cursor with the trackball over a point, and toggles the blue switch. The selected point will be marked in red. If there are several points

The user is then requested to enter a title for the plot. The default title is the input file name, but the user may change the title to any character string (maximum length of 40 characters).

After the selected plot is finished, the following menu will appear on the VT100 screen:

```
TYPE <CR> - TO ZOOM, SCROLL AND IDENTIFY
               POINTS WITH THE CURSOR
      New - TO CREATE NEW PLOT
    Print - TO GET A HARD COPY OF PLOT
   Robust - TO INVOKE ROBUST ESTIMATION ROUTINES
     Quit - TO HALT PROGRAM EXECUTION
```

When an option is selected, only the first letter of the user's response is examined. Hence, typing any character string that starts with P will produce a hard copy of the plot. A description of each option will now be given

<CR> - Typing the RETURN key will allow the user to zoom, scroll, and select points using the cursor. Further details will be presented next.

New - By typing New (or N), the user can create a new plot on the graphics terminal. The user will be prompted for the type of plot, and data field. As before, default values are available for the various input prompts.

Print - Typing Print (or P) will cause the statistical plot on the graphics terminal to be printed on the line printer. The user will then be asked to select again from this same menu.

Robust - Entering Robust (or R) will allow the user to perform the robust estimation techniques on the current set of plotted data. Further details on the robust estimators will be given later.

If the input file is on the same directory as the user, only the filename and type need to be specified. The default file type is DAT, and if no version number is given, the file with the highest version number is used. Volume 2A of the VAX/VMS Reference Manuals contains a complete description of file specification.

If the word QUIT is typed for the input file name, the statistical plot program will stop execution and return control to VMS. The user should note that if the word QUIT is typed for many of the input prompts, the program will terminate and return control to VMS.

The program will first display the latitude and longitude limits of the input gravity survey file, along with the number of points in the file. The user will then be prompted for the type of plot with

```
WHAT TYPE OF PLOT?
TYPE 1 FOR QQ PLOT
     2 FOR ECDF PLOT
 CURRENT VALUE IS            1
*****ENTER NEW VALUE OR <CR>*****
```

The default, as shown, is the QQ plot. The following message will then appear on the screen:

```
WHAT VALUES PLOTTED?
ENTER 1 - ELEVATION
      2 - FREE-AIR ANOMALY
      3 - BOUGUER ANOMALY
 CURRENT VALUE IS  3
*****ENTER NEW VALUE OR <CR>*****
```

The default value is a plot of the Bouguer anomaly data.

interactively from one of the VT100 terminals, and it requires the user to respond to several prompts and menu selection options.

2.7.2 Program Limitations

Since the statistical plot program requires the use of the Lexidata graphics terminal, the user should make sure that the graphics terminal is available before running the program. The user should also make sure that the switches located on the trackball unit are in the OFF position.

The statistical plot program handles files with up to 5000 points. The robust estimators can also handle the same amount of data, except for the Hodges-Lehmann estimate which is limited to 1000 points. Larger files should be separated into smaller subfiles before processing with this program.

2.7.3 Running the Statistical Plot Program

To run the statistical plot program, the user enters the following command:

```
RUN STATPLOT
```

The user will then be prompted to enter the file name containing the gravity survey data to be examined. The file name must include all qualifiers to identify the file uniquely. A complete file specification has the following format:

```
device:[directory]filename.type;version
```

The punctuation marks (colons, brackets, periods) are required to separate the various components of the file specification.

After the user enters the zoom level, the plot will be enlarged at the specified level. The program is still in the scroll mode, so that the picture can be moved about the screen. The blue switch can be toggled again, to enter a different zoom level. To freeze the screen and terminate the zoom and scroll option, the user should toggle switch A (i.e., the left-most white switch) on the track-ball unit.

2 - Typing a 2 will prompt the user for different viewing angles. The current angles are now the default values, which the user may specify by typing the RETURN key. After the plot is redrawn, the first menu selection will appear on the screen.

3 - By typing a 3, the user can create a new plot on the graphics terminal. The user will be prompted for new latitude and longitude limits, a different data field, a new title, and new viewing angles. As before, default values are available for the various prompts.

99 - Typing a 99 will cause the program to terminate.

2.7 ROBUST ESTIMATION AND STATISTICAL PLOT PROGRAM

2.7.1 Purpose and Scope

The purpose of the statistical plot program is to display on the Lexidata graphics terminal both quantile-quantile (QQ) plots and empirical cumulative distribution function (ECDF) plots of a specified data field from a file containing gravity survey data. The program also contains routines for performing robust estimations on the data displayed. The program can also invoke the cursor to identify individual points or subsets of points. The statistical plot program is run

<CR> - Typing the RETURN key will allow the user to zoom, scroll, and change the viewing angle. Further details will be presented next.

Replot - By typing Replot (or R), the user can create a new plot on the graphics terminal. The user will be prompted for new latitude and longitude limits, a different data field, a new title, and new viewing angles. As before, default values are available for the various prompts.

Print - Typing Print (or P) will cause the surface plot on the graphics terminal to be printed on the line printer. The user will then be asked to select again from this same menu.

Quit - Typing Quit (or Q) will cause the program to terminate.

If the user selects the zoom, scroll, and change viewing angle option, the following menu will appear on the screen:

```
ENTER 1 - ZOOM AND SCROLL
      2 - CHANGE VIEWING ANGLE
      3 - NEW PLOT
     99 - TO EXIT FROM THE PROGRAM
```

A decription of each of the options will now be presented.

1 - By typing a 1, the user invokes the zoom and scroll option. By rolling the trackball, the user can move the surface plot around the screen. To invoke the zoom feature, the user should move the area to be enlarged to the upper left corner of the graphics screen, and toggle the blue switch on the trackball. The user will then be prompted for the zoom factor at the VT100 terminal. The zoom factor is any integer value between 1 and 16.

The user should respond with the name of one of the files containing points selected as lying along a track. If the word QUIT is typed for the input file name, the program will stop execution and return control VMS.

After the file name is entered and the track file is read, the VT100 terminal screen will clear, and a list of all the source numbers which occur in the file will be displayed, along with the number of points associated with each source number. The user will then receive the following prompt at the terminal:

```
ENTER SOURCE NUMBER OF RECORDS TO BE SAVED
ENTER A SOURCE NUMBER <0 TO TERMINATE PROGRAM
```

The user should respond with the source number of the points to be kept in the file. Points with a different source number will be deleted from the file. The user should note that the word QUIT is not an acceptable response to this prompt. To terminate the program, the user should enter a negative number for the source number.

The program will create a new file with the same name as the input file. The new file will contain only those points with the specified source number, and the records in the file will be sorted by latitude and longitude. The original file will still exist on the disk.

Program SPLINE

The purpose of program SPLINE is to compute the value of the free-air anomaly along a track at an intersection point. In order for this program to function properly, the user must first run CLEANUP. To run the SPLINE program, the user should type:

RUN SPLINE

A message will appear on the screen warning the user to run CLEANUP before using program SPLINE. Next, the user will receive the following prompt at the terminal:

```
ENTER INPUT FILE NAME OR
      QUIT TO HALT PROGRAM EXECUTION
```

The user should repond with the name of a file containing track data processed by the program CLEANUP. If the word QUIT is typed for the input file name, the program will stop execution and return control to VMS. The user should note that if the word QUIT is typed for many of the input prompts, the program will terminate and return control VMS.

Next, the user will receive the following prompt at the terminal:

```
ENTER LATITUDE AND LONGITUDE OF INTERSECTION PONT
ENTER A VALUE > 90 FOR THE LATITUDE
      TO TERMINATE THE PROGRAM
```

The user should note that the word QUIT is not an acceptable response for this prompt. To terminate the program, the user should enter a value larger than 90 for the latitude.

After the user enters values for the latitude and longitude, the program will echo the values and ask if they are correct. In case of an error, the user can reenter different values for latitude and longitude. If the latitude and longitude coordinates are correct, the program will compute the free-air anomaly value of the track at the intersection

point and display the value on the VT100 screen. The user can then repeat the procedure and enter the coordinates of a different intersection point for the same track.

Program LINPRO

The purpose of the program LINPRO is to set up and solve the linear programming model corresponding to the previously identified tracks and intersection points. To execute this program, the user should type:

```
RUN LINPRO
```

The user will first be asked whether the results of this program are to be printed on the line printer, and will be prompted for a title to be printed at the top of the output. Next, the user will receive the following prompt:

```
IS THE LINEAR PROGRAM ALREADY STORED ON A FILE ?
ENTER Yes OR No
```

For the initial processing of a given set of intersection points, the user should respond with "NO". Details for this option will be explained shortly.

Next, the program will prompt the user for the number of tracks and the number of intersection points. The number of tracks must be greater than 1 and less than 26, and the number of intersection points must be greater than 0 and less than 51. For the first intersection point, the user will receive the following prompt:

```
ENTER TRACK NUMBER AND CORRESPONDING VALUE FOR THE
  TWO TRACKS WHICH MEET AT INTERSECTION POINT 1

ENTER A 0 (ZERO) FOR THE TRACK NUMBER TO TERMINATE
  THE PROGRAM
```

The user will then be prompted for the two track number and their corresponding free-air anomaly values at the intersection point. At any intersection point, the two track numbers must be different (i.e., a track cannot intersect itself). The user should also note that the word QUIT is not an acceptable response for this prompting sequence. To terminate the program, the user should enter a zero for either of the track numbers.

After the track numbers and free-air anomaly values have been entered, the program will echo the values and ask the user to make any required changes. An example of a typical display seen on the VT100 screen is:

INTERSECTION	TRACK	VALUE
1	4	-10.2
	5	-17.6

The user now has the option of changing either of the track numbers and/or their corresponding values.

The program will repeat this prompting sequence for each of the intersection points. After initially entering the track numbers and free-air anomaly values, the user will have the option of making any changes and/or corrections. When the user is sure that all the intersection points are correct, the following message will appear on the VT100 screen:

SAVE THIS LINEAR PROGRAM ?
ENTER Yes OR No

The user now has the option of saving all the information regarding the the intersection points and corresponding track numbers and free-air anomaly values in a data file. This is a very useful feature in that once this data set has been saved, the user can run the linear program model over again, using the same tracks and intersection points, without having to retype all the numbers. If the user decides to save the linear program, there will be a prompt for a file name under which all the data will be stored.

To recall this linear program model, the user should respond with a "YES" to the previously asked question of whether the linear progam was already stored on the file. There will be a prompt for the file name where the data are stored. After the linear program is read in, the user will have the opportunity to make changes. The altered linear program can be saved in a different file.

The program will then invite the user to place any desired constraints on the individual track adjustments. In this step, the user must rely on judgment and experience in assigning a numerical value to the quality of each track. For example, the adjustment to a high-quality source might be restricted to values below 1 mgal. On the other hand, for a poor-quality source, the adjustment might be as much as 20 mgal. The adjustment for tracks with no constraints can assume arbitrary values.

If the user decides to put constraints on the track adjustments, the following messages will appear on the VT100 screen:

```
THE CONSTRAINTS ARE OF THE FORM : ABS(A(I)) .LE. C

   WHERE A(I) = ADJUSTMENT APPLIED TO TRACK I
            C = POSITIVE REAL NUMBER, OR 0.0

ENTER TRACK NUMBER, I, AND CONSTRAINT VALUE, C

WHEN YOU ARE FINISHED ENTERING CONSTRAINTS
     ENTER -1 FOR THE TRACK NUMBER
```

So, if the user wants the absolute value of the adjustment for track number 3 to be restricted to 5.0 mgal or less, there should be entries of 3 for the track number and 5.0 for the constraint value. The program will echo the track number and the constraint, and ask the user if these are correct. The user can then proceed to enter more track numbers and constraints until a -1 is typed for the track number.

The program will then solve the linear programming problem, and the results will be displayed at the terminal. The results will also be printed on the line printer if the user has previously requested it.

2.8.4 Running the Ocean Track-Crossing Adjustment Programs

This section will present a step-by-step instruction guide on how to combine the four previously described programs to solve the track-crossing adjustment problem.

The first step is for the user to run the program SCATTER on a selected file of ocean data. The user should then use the cursor to select the various tracks. Each track should be written into a separate file. The file names given to the tracks should be similar so that the user can easily identify each track. For example, all the track files could start with the letters TRK, in which case file name TRK4475

will contain the data points for the track with source number 4475. If there are two tracks with the same source number, the user can use trailing letters on the file name (e.g., TRK4475A and TRK4475B).

The user should now run the program CLEANUP on the selected track files to remove any extra data points and to sort the file by latitude and longitude. Following this step, there will be two versions of each of the track files. To remove all the old versions, the user should type PURGE.

Next, the user should create a master file containing all the preprocessed track files. To accomplish this task, the user can use the COPY command. If the user has specified all the track files with unique identifiers, as was suggested earlier, the following command will copy all the track files into a master file.

```
COPY TRK*.DAT MASTER.DAT
```

This command will copy all files whose file names begin with the letters TRK into a file named MASTER.DAT. If the track files were not created in this manner, the user can copy the individual files into a master file with the following command

```
COPY file1,file2,....,fileN MASTER.DAT
```

Next, the user should again run the program SCATTER using the master data file. This plot will clearly show all the selected tracks and their intersection points. The user should obtain a hard copy of this plot. Note that the user must exit from the program SCATTER before the plot is sent to the line printer.

Now, the user should have available a copy of the ocean track-crossing adjustment worksheet, which is used to record the locations of the intersection points. A sample worksheet is shown in Fig 2.8-1. The user should again run program SCATTER using the master data set. With the cursor, the user determines the latitude and longitude coordinates of the intersection points, and records them on the worksheet. The user should also mark the intersection point number on the hard copy of the scatter plot for later reference.

When all the intersection points have been identified, the user should run the program SPLINE using the individual track files as the input files. It is important to note that these individual track files MUST be preprocessed by the program CLEANUP before they can be used by the SPLINE program. The SPLINE program will determine the free-air anomaly values at the intersection points. Again, the user should use the track-crossing adjustment worksheet to record the free-air anomaly values at the various intersection points.

Next, the user must assign track numbers to the selected tracks. Numbering of the tracks should begin with one and proceed sequentially. If each track has a unique source number, then the user can number the tracks consecutively. However, if there are different tracks with the same source number, the user has an option. If the user wants the same adjustment to apply to all tracks with the same source number, then all such tracks should have the same track number. However, if the tracks are to have different adjustments, they should be given different track numbers.

Finally, the user is ready to run the progam LINPRO to find the adjustment factors. The first time the user runs this program, it is necessary to type in the intersection points

Intersection Point	Latitude	Longitude	Source Number	Track Number	Value

Figure 2.8-1 Ocean Track-Crossing Adjustment Worksheet

and the corresponding free-air anomaly values. All of the numbers to be typed in can be read off the worksheet. Once the intersection points and free-air anomaly values have been initially entered, they can be saved in a data file. This file can then be recalled at any time, and the user can make additions and/or corrections as desired.

2.9. ERROR AND DIAGNOSTIC MESSAGES

2.9.1 General Error Messages

The programs which make up the gravity data evaluation software have automatic error traps to flag erroneous data supplied by the user. When the user is prompted to select an option from a menu, the programs check the validity of any selected option, and reprompt the user if an invalid option has been requested. If latitude and longitude values are entered by the user, the programs will check for consistency and reprompt the user if some of the values are suspect.

As was stated in the previous sections, the gravity input file name has no default value. The input file name MUST be specified by the user. If an improper file name is entered, the program will display one of the warning messages listed in Table 2.9-1 on the VT100 screen.

All three of these messages are caused by nonfatal errors which are trapped by the program. In each case, the user is reprompted to enter a new input file name.

There is one fatal error which will cause each of the programs to terminate execution. If an error occurs while the program is reading the input gravity file, the following message will appear on the screen:

TABLE 2.9-1

WARNING MESSAGES

MESSAGE	POSSIBLE CAUSES
COULD NOT FIND FILE: filename CHECK SPELLING	The program could not find the input file specified by the user. Make sure the file name is spelled correctly and all necessary qualifiers are included in the file name.
ERROR IN OPENING FILE: filename	The program could not open the input file for reading. Could happen if another user is also reading same file. Wait until file is released, or use a different file.
FILE SPECIFICATION ERROR	User has entered an invalid file name. Possible causes of invalid file names are too many characters (maximum of 9) or illegal characters in file name.

AN ERROR OCCURRED WHILE READING FILE : filename
CHECK FILE FOR PROPER INPUT FILE FORMAT

The user should examine the input file to make sure that the records in the file are in the WSS online format (see Section 2.2).

Some of the programs also have errors specifically related to the individual program. These are the scatter plot program, the statistical plot program, and the ocean track-crossing adjustment programs. A description of these errors will now be presented.

2.9.2 Scatter Plot Program Error Messages

The scatter plot program contains a fatal error message that involves the system plot routine SUPMAP. This routine is called to draw political boundaries and coastlines which may lie within the limits of the scatter plot. If an error occurs in this routine, the following message will appear on the VT100 screen:

```
ERROR IN SYSTEM PLOT ROUTINE SUPMAP
CHECK PLOT REFERENCE MANUAL  P. 147
FOR ERROR NUMBER IER
```

where IER = error number

The plot reference manual (Ref. 7) has a list of the various error flags and their explanation. The most common error is caused by improper latitude-longitude limits of the plot.

2.9.3 Statistical Plot Program Error Messages

In addition to the errors mentioned above, there are other nonfatal errors in the robust estimation. If there are not enough points to do the robust estimation computations, the following message will appear on the VT100 screen:

```
*** ERROR: NOT ENOUGH POINTS IN BIWEIGHT
```

Many of the robust estimators cannot function properly with small amounts of data. The user should provide more data for the estimators.

Another error in the robust estimators may occur in the M-estimate and sine-estimate if the algorithm does not converge after 25 iterations. The message presented will be of this type:

```
***** WARNING: SINE-ESTIMATE DOES NOT CONVERGE AFTER
25 ITERATIONS
```

This will occur if there are a small number of widely scattered data points.

2.9.4 Ocean Track-Crossing Adjustment Programs Error Messages

For the program CLEANUP, the following message is due to a fatal error in the sort routines.

```
FATAL ERROR IN SORTING ROUTINES
```

Since the error occurred in the system sorting routines, there could be a problem with the system software. The user should consult the systems manager if this error arises.

The following error message may appear on the VT100 screen while executing the program SPLINE:

```
AN ERROR OCCURRED IN THE IMSL ROUTINE WHICH PERFORMS
   THE SPLINE FIT.  LOOK IN THE IMSL DOCUMENTATION
   UNDER THE ROUTINE NAMED :  ICSSCV
   FOR ERROR NUMBER IER
```

where IER = error number

This error is probably caused by duplicate points in the track file. The user should run the program CLEANUP on the track file, and then try the program SPLINE again.

When the user is running the program LINPRO, the following error message may appear on the VT100 screen:

```
AN ERROR OCCURRED IN THE IMSL ROUTINE WHICH SOLVES
   THE LINEAR PROGRAM.  LOOK IN THE IMSL DOCUMENTATION
   UNDER THE ROUTINE NAMED : ZX3LP
   FOR ERROR NUMBER  IER
```

where IER = error number

his error could occur if the user specifies too many tracks nd/or intersection points. If the linear program problem is ery large, it should be separated into smaller sections.

never the program asks you a question, you may terminate program by typing

QUIT

program starts by prompting for the name of the master a file that you wish to use:

ENTER INPUT FILE NAME

may respond by entering RETURN to select the default file ne displayed on the VT100 monitor, or you may type a file ne of your choice.

The program responds by displaying the prompt

ENTER NUMBER OF RECORDS IN A BATCH

may enter RETURN, which chooses the default value displayed the VT100 monitor, or type a positive integer. This integer be any integral power of 2 in the range from 2 to 4096, or may be a highly composite number, such as 500, 1000, etc. you choose a number that is not highly composite, such as), then the program will fail when the periodogram computation attempted. (This requirement for composite batch sizes is nerent in the FFT algorithm.) The number of records in a batch the number of consecutive data pairs from channels 1 and 2 the master data file that are used to compute each periodogram. larger this number, the greater the number of frequency bins the periodogram.

The program continues by displaying

ENTER NUMBER OF BATCHES

These choices are self-explanatory except for choice F, which allows you to recompute the power spectra and coherence with a bandwidth and frequency resolution of your choice. The F option allows you to specify a minimum frequency limit, a maximum frequency limit, and the number of different frequencies at which the spectra and coherence are to be evaluated between these limits. The P, C, and F options cause the program to ask for minimum and maximum frequency limits of your choice. You may invoke autoscaling for these limits as follows: type 1 for both minimum and maximum frequency limits -- the program will respond by autoscaling the frequency scale. This technique of typing 1 for both minimum and maximum limits also invokes autoscaling when the program asks for y-axis limits in the P and C options.

The original time series data are not available for replotting once the power spectra have been are computed.

3.5 PERIODOGRAM SPECTRUM ANALYSIS AND PLOTTING (PROGRAMS FFT AND PLOTFFT)

Any master data file having the format described in section 3.2 can be analyzed by running program FFT. This program estimates the power spectrum of the data by computing an averaged matrix periodogram from selected batches of windowed data in the master data file. Program FFT automatically saves the periodogram and its standard deviations in a disk file. After program FFT has created the periodogram file, you may plot the auto spectra and squared coherence on the Lexidata graphics terminal by running program PLOTFFT.

To run program FFT, type

```
RUN FFT
```

It is usually advisable to choose Y; later in the program you have the choice of overriding the autoscaling mode. (For illustrative purposes it is assumed that autoscaling is chosen.)

The program continues by computing the complex spectral density matrix for the best AR model. This matrix is computed at 300 logarithmically-spaced frequencies spanning the range from 0.3 times the reciprocal of the data length to the folding frequency (half the sampling frequency of the data being analyzed).

The program then continues by plotting the autospectra for both data channels on the Lexidata graphics terminal. The program displays the usual three-choice menu to give you the opportunity to print copies of the plots on the Printronix printer/plotter.

Entering RETURN or typing CL clears the screen. The program then continues by plotting the spectral coherence between the two channels of data, and the usual menu is displayed on the VT100 monitor. You may then choose to print the coherence function on the Printronix printer/plotter.

You may now replot the autospectra or the coherence with linear or logarithmic scales of your choice. This is done by entering RETURN or typing CL, causing a new menu to appear on the VT100 monitor:

P - PLOT POWER SPECTRUM

C - PLOT COHERENCE

F - CHANGE THE FREQUENCY RESOLUTION

Q - QUIT (terminates the program)

You have the choice of entering RETURN, which chooses the default value for model order, or you may type a number in the range of 1 through 40. The larger the order selected, the longer the computations take for fitting the models to the data. A maximum model order of 20 is reasonable for many data sets.

Once the maximum AR model order is selected, the program subtracts the means from both channels of data and then fits a family of AR models to them. The Akaike Information Criterion (AIC) is computed for each model and displayed on the VT100 monitor. The model order for which the AIC attains its algebraically smallest value is selected by the program. This is the model order that is best supported by the data for the purposes of modeling the process that generated the data. The program continues by plotting the sum-squared one-step-ahead prediction errors of the selected model for each data channel. If these plots are nearly straight lines, then the data have uniform statistical properties. On the other hand, if there are sudden large deviations from a straight line in these plots, then the locations of these deviations show places in the data set where there is exceptional behavior that is atypical of the rest of the data. These plots are useful for identifying outliers in nonhomogeneous data sets.

The menu described previously is now displayed; you have the option of continuing the program, printing the color graphics plot on the Printronix printer/plotter, or ending the program. When you continue the program by entering RETURN or typing CL, the following prompt appears on the VT100 monitor:

DO YOU WANT AUTO SCALING OF PLOTS?

You may respond by typing either Y or N.

terminal. A caption is plotted at the bottom of the display. The first line of the caption is the header from the master data file that was used in creating the data set currently being analyzed. The second and third lines contain the following information

- Decimation Factor (used in smoothing and resampling of data)
- Data Set (name of data file currently being analyzed)
- Number of Records (number of data samples currently being analyzed)
- Ramps (tells whether or not ramps were removed from the data and lists their slopes in data units per sample).

The fourth line in the caption contains any information that you previously typed into the header when the current data set was created by program GETDATA. This four-line caption will appear at the bottom of each plot generated by program AR.

After the program is finished plotting the data, it displays a menu on the VT100 monitor. The three options in this menu have the following meanings

<cr>/CLear - clears the Lexidata screen and continues the program

Print - prints a copy of the Lexidata plot on the Printronix printer/plotter

Quit - terminates the program

After you enter RETURN or type CL, the program displays the prompt

MAXIMUM AUTOREGRESSIVE MODEL ORDER (<41)

GETDATA are valid. In typing the name of the desired file, the suffix ".DAT" may be omitted; e.g., BELL1 is a valid file name. (There are certain reserved system names that can not be used as file names. You may wish to consult your systems manager for information about restrictions on data file names. Examples of invalid file names for this program are QUIT and EXIT.) Entering RETURN selects the default filename.

After reading the data from the selected file, the program displays the prompt

DO YOU WANT RAMPS REMOVED?

(You always have the option of terminating the program when a yes-no type questions like this one is asked; just type QUIT RETURN.)

To continue the program, you may respond by entering RETURN, which selects the default choice displayed on the VT100 monitor, or you may type Y or N If you respond with Y, then a straight line is fitted to each channel of data by least squares and subtracted; the straight line is constrained to have an average value of zero so that the arithmetic means of the data in each channel are unchanged. The slopes of the straight lines are listed on the VT100 monitor and also listed in the captions of all graphs. The units of the slopes are data units per sample.

The program next displays the prompt

DO YOU WANT A PLOT OF THE DATA?

You may respond by typing Y or N. If you respond with Y, then both channels of data will be plotted on the Lexidata graphics

of each plot. After you type the plot header, enter RETURN. The program then creates the new file of smoothed and resampled data.

The next prompt asks if you want to start all over again and select more data from a master data file. You may respond with Y or N (or YES, NO). Typing Y causes the program to start over again. Typing N causes the program to terminate.

3.4 AUTOREGRESSIVE SPECTRUM ANALYSIS (PROGRAM AR)

Any data file created by GETDATA may be analyzed by running program AR, which plots the data, subtracts the mean from each channel of data and fits a family of autoregressive (AR) models to the residuals, selects the best model via the Akaike information criterion, and uses this model to generate prediction error, power spectrum, and coherence plots. The program AR is interactive and prompts the user at each step. In the following discussion, it is assumed for illustrative purposes that a data file named BELL1.DAT has previously been created by program GETDATA.

It is recommended that you start by typing DIR *.DAT RETURN. This causes a list of the data files in the current library to be displayed on the VT100 monitor for easy reference. For the present example, BELL1.DAT would be among the file names listed on the monitor. (If you do not want a list of the data files displayed on the VT100 monitor, then this step should be omitted.)

To run program AR, type RUN AR. The program AR will then respond with a prompt asking for the name of the data file that is be to analyzed. Data files created by program

automatically by the program to save your resampled data set for later analysis by the program AR. If you do not plan to use AR, then there is no need to use this file-creating option. The rest of this paragraph describes how to proceed should you wish to save your resampled data set. Enter RETURN, to select the default file name displayed on the VT100 terminal, or type a file name of your own choice. Examples of valid file names are BELL1 or BELL1.DAT. (The program will append the suffix ".DAT" automatically if you omit it.) The program then displays a header that becomes the first part of the new data file. The first line of this header is a copy of the header in the master data set. The remaining lines list the decimation factor, the name of the file to be created, the number of data samples in the new file, the sampling rate of the smoothed data, the location of the first datum taken from the master data file in creating the current data set, and information about any linear trends (ramps) that might have been removed from the resampled data (the slopes of linear trends are expressed in data units per sample). You are now given the opportunity of typing a single line of additional information about the smoothed and resampled data. The maximum length of this line is 72 characters. Avoid using the dollar sign "$" because this symbol will cause all others following it to be ignored during certain printing operations. The valid characters are letters of the alphabet, integers, comma, period, and the following special symbols

+ - * / ( ) =

You may use the DELETE key to correct typing errors before pressing RETURN. After RETURN is pressed, the program asks for a plot header. This is a short string of characters (length ≤ 22 characters) that will be printed automatically at the top

The program next generates plots of the smoothed and resampled data on the Lexidata graphics terminal, provided that the Lexidata terminal is available. The plot is for channel 1.

A menu is now displayed on the VT100 monitor; you may select any of the three possibilities listed there by typing the command you select. As a shortcut, you may type only the capitalized portion of the command. In this menu, <cr> is an abbreviation for RETURN. The commands are used for the following purposes:

<cr>/CLear - to clear the Lexidata screen and continue to the plotting of the data in channel 2

Print - to print a copy of the graph displayed on the Lexidata screen

Quit - to terminate the program

The abbreviations CL, P, and Q are valid abbreviations for CLEAR, PRINT, and QUIT.

After the data for channel 2 have been plotted on the Lexidata terminal, the menu is again displayed. This time when <cr>/CLear is selected, the program responds by asking whether you wish to save the data in a disk file. You may respond by typing YES, NO, or QUIT. The abbreviations Y, N, and Q are valid.

Typing Q will terminate the program. Typing N will cause the program to ask if you wish to continue by selecting some more data from the master data set (typing NO or N in response to this question will terminate the program). Typing Y will cause the program to ask for the name you want to use for the output file. This is the file that will be created

of either entering RETURN to select the default value for the decimation factor, or you may type a number of your choice.

The next prompt asks you for the sampling interval. This is the number of seconds between adjacent samples in the master data set. If you do not know the correct value for this quantity, you may type 1; the program will then continue to run and you will be able to view the data, but the header information that is generated for the selected data may be incorrect. You have the choice of either entering RETURN to select the current default value of the sampling interval, or you may type a number of your own choice.

The next prompt asks for the name of the dimensional units for the data being analyzed. For example, the correct response to this prompt for gravity gradiometer data is EOTVOS. As usual, you may simply enter RETURN to select the current default value for data units displayed on the VT100 monitor.

The program now responds by reading the master data file, listing the ASCII header from the master file, and printing the number of smoothed and resampled data points that have been produced during the reading operation. The number of resampled data points may be smaller than the number you requested; this happens when there are not enough data in the master data file to satisfy your request.

The next prompt asks whether you want linear trends removed from each channel of the processed data set. Responding with Y (or YES) to this prompt causes a best-fit least-squares straight line of zero mean value to be subtracted from each channel of resampled data. You may respond by entering RETURN for the current default setting, or you may type either N (or NO) to defeat the straight-line subtraction.

3.3 DATA SELECTION (PROGRAM GETDATA)

To select data from a master data set, type RUN GETDATA RETURN. You will be asked for the name of the file containing the master data set. There are now three courses of action available to you:

1. Type RETURN to choose the current default file
2. Type the name of a file using standard VAX format
3. Type QUIT to terminate the program. You may use this command whenever the program asks you a question.

You will now be asked for the number of smoothed data samples you wish to generate from the master data set. You may either enter RETURN to select the current default value displayed on the VT100 monitor, or you may type in a number of your choice followed by RETURN.

The next prompt asks for the location of the first datum you wish to select from the master data set. Typing 1 chooses the first datum in the master data set. You have the choice of either entering RETURN to select the current default value for the location, or typing a number of your choice.

The next prompt asks for the decimation factor. This number controls the degree to which the data are to be smoothed and resampled. For example, a decimation factor of 1 results in no smoothing and no resampling, while a value of 10 produces a selected set of data in which each sample is the average of 10 samples in the master data set. Therefore, in this case the sampling rate in the smoothed data set is one tenth the rate in the master data. You have the choice

3.2 DATA FORMATS

The programs are currently set up for two-channel data sets. This means that there should be two data values at each sample time in the master data set. With GETDATA, a maximum of 5000 two-channel samples can be selected for viewing from the master data set at one time.

The master data set should be a file with the following structure:

First Record = ASCII header describing the data set

(length equal to or less than 80 characters)

Example: BELL AF GGI S/N2 UMB ANGLE RUN 06/20/80

Other Records = pairs of data points in BINARY (2A4) format

2nd Record = 1st datum pair for channel 1 and channel 2

3rd Record = 2nd datum pair for channel 1 and channel 2

4th Record = 3rd datum pair for channel 1 and channel 2

5th Record = 4th datum pair for channel 1 and channel 2

*

*

*

The name of the master data set should preferably end in the suffix ".DAT"; e.g., "BELL1.DAT" is a valid data file name. (There are certain reserved system names that cannot be used as file names. You may wish to consult your systems manager for information about restrictions on data file names.)

GETDATA and AR programs and yields autoregressive spectrum estimates. The second choice is to create a file of periodogram spectrum estimates by running program FFT and then to plot the spectrum estimates by running program PLOTFFT. This second choice uses the FFT and PLOTFFT programs and yields smoothed periodogram spectrum estimates.

The program AR computes an optimal autoregressive model for the process that generated the data and uses this model to plot the following quantities:

- Data being analyzed
- Sum-squared prediction errors of the AR model
- Autospectra for the two data channels
- Spectral coherence between the two data channels.

The program FFT smooths selected batches of data from a master data set, windows the batches to reduce spectral leakage, computes a periodogram spectrum estimate, and saves the results in a disk file for later use by the periodogram plotting program PLOTFFT.

The program PLOTFFT uses the data file created by FFT to plot the following quantities:

- Autospectra for the two data channels
- Spectral coherence between the two data channels.

3. USER'S GUIDE TO THE SMOOTHING AND SPECTRUM ANALYSIS PROGRAMS

3.1 INTRODUCTION

There are four programs for the smoothing and spectrum analysis of time series data:

- Data smoothing and selection program GETDATA
- Autoregressive spectrum analysis and plotting program AR
- Periodogram spectrum analysis program FFT
- Periodogram spectrum plotting program PLOTFFT.

These programs are currently set up for analyzing two-channel time series (such as test data for gravity gradiometers that have two output channels). The program GETDATA is used to view selected portions of a master data set on the interactive graphics terminal. In addition, this program allows the user to prepare special data sets for later analysis via the program AR; in particular, the user may use GETADATA to smooth selected portions of a master data set, to resample the smoothed data at a selected rate, to view the results on the graphics terminal, and to save the smoothed and resampled data in a new file on disk.

After using the program GETDATA to plot data for analysis, the user has two choices for futher analysis of these data. The first choice is to create a file of selected data by using program GETDATA and then to do spectrum analysis of these data by running program AR. This choice uses both the

You may enter RETURN to choose the default number of batches, or type an integer of your choice. The program will attempt to segment the master data file into as many nonoverlapping adjacent batches as you specify. If there are insufficient data, the program forms as many batches as it can from the available data. These batches will contain the number of records you requested for each batch.

The program continues by displaying

ENTER STARTING RECORD

You may enter RETURN to select the default record, or type an integer. For example, if you type 1000, the program will skip the first 999 pairs of data in the master data file and start the first batch with record 1000. (Plots of the master data set can be generated interactively by running program GETDATA before you run program FFT.)

The program continues by displaying

ENTER DECIMATION FACTOR

You may respond by entering RETURN to choose the default decimation factor, or typing an integer. The value 1 results in no decimation (no smoothing and resampling). A value greater than unity causes the program to smooth the master data with a running mean and then to resample these smoothed data. The decimation factor is equal to the number of data points in the running mean, and it is also equal to the factor by which the sampling interval in the master data set is increased by the resampling process.

The program continues by displaying

DO YOU WANT A RAMP REMOVED FROM EACH BATCH?

Entering RETURN selects the default option. Typing Y or N selects YES or NO options. Selecting option YES causes the program to subtract a zero-mean least-squares straight line from each channel of data. These lines are computed for each batch of data.

The program continues by displaying

ENTER KAISER-WINDOW ALPHA PARAMETER

You may enter RETURN to select the default value, or type a real number greater than or equal to zero. The Kaiser window is a bell-shaped weighting function. The program multiplies each batch of data by this weighting function to reduce spectral leakage in the periodograms. A value of zero for the alpha parameter yields a rectangular window (constant weighting). The larger the alpha parameter, the greater the attenuation of spectral leakage. However, larger values of alpha also yield lower frequency resolution in the periodogram. A value of 5.4414 has given satisfactory results in the analysis of gravity gradiometer self-noise data. The Kaiser window is defined in Ref. 4.

The program continues by asking for the number of seconds between adjacent samples in the master data set

ENTER SAMPLING INTERVAL

You may enter RETURN, which chooses the default sampling interval displayed on the VT100 monitor, or type a positive real number of your choice. If you do not know the correct sampling interval, you may type 1. The program will then continue to execute, but the spectral plots will be incorrectly labeled.

The program continues with the prompt

ENTER FRACTIONAL BANDWIDTH

The fractional bandwidth is a real number greater than or equal to zero that determines the frequency resolution of the spectrum estimates after the averaged periodogram is smoothed with a running mean. The width of the running mean is a fraction of its center frequency, and this fraction is the fractional bandwidth. This method of smoothing the periodogram yields constant-percent-bandwidth resolution, which is appropriate for the logarithmic frequency scales that are normally used for plotting spectra.

The program continues by processing the specified batches of data. During this processing, the means are subtracted from each windowed batch and then periodograms for each batch of data are computed with a mixed-radix Fast Fourier Transform (FFT) algorithm. These periodograms are averaged to produce a mean periodogram; the standard deviations of the periodograms are also computed if more than one batch of data is available.

The program continues by asking for the name of the file in which you wish to save the mean periodogram and standard deviations:

PLEASE ENTER THE NAME OF THE OUTPUT FILE

You may enter RETURN to choose the default file name, or type a file name of your choice. Examples of valid file names are BELL1 or BELL1.DAT. (The program will append the suffix ".DAT" automatically if you omit it.)

The program continues by displaying a header that becomes the first part of the new data file. The first line of this header is a copy of the header in the master data set. The remaining lines list the decimation factor, the name of the file to be created, the number of data samples in the new file, the sampling rate of the smoothed data, the location of the first datum taken from the master data file in creating the current data set, and information about any linear trends (ramps) that might have been removed from the resampled data (the slopes of linear trends are expressed in data units per sample). You are now given the opportunity of typing a single line of additional information about the smoothed and resampled data. The maximum length of this line is 72 characters. Avoid using the dollar sign "$" because this symbol will cause all others following it to be ignored during certain printing operations. The valid characters are letters of the alphabet, integers, comma, period, and the following special symbols:

    + - * / ( ) =

You may use the DELETED key to correct typing errors before pressing RETURN. After RETURN is pressed, the program asks for a plot header. This is a short string of characters (length equal to or less than 22 characters) that will be printed automatically at the top of each plot. After you type the plot header, enter RETURN.

The program continues by creating the new data file on disk and then displaying

    DO YOU WANT ANOTHER REGION OF DATA?

This allows you either to terminate the program by typing N, or to start the program over again by typing Y. You may alternatively enter RETURN to select the default option displayed on the VT100 monitor.

To plot the periodogram spectra on the Lexidata graphics terminal, you should run program PLOTFFT by typing

```
RUN PLOTFFT
```

Whenever the program asks you a question, you may terminate the program by typing

```
QUIT
```

Program PLOTFFT starts by displaying

```
ENTER INPUT FILE NAME
```

Any file created by program FFT is valid. You may either enter RETURN to select the default file name, or you may type a file name of your choice.

The program continues by displaying a four or five line header that describes the data set selected for analysis. The first line is the header from the master data set from which the current data set was derived. The second line gives the decimation factor and the name of the current data set. The third line lists the number of batches, number of samples per batch, the sampling interval, and the location (NSTART) of the first datum taken from the master data file in creating the current data set. The fourth line lists the Kaiser-window alpha parameter and the fractional bandwidth. The last line may be blank or may contain a descriptive message about the data.

The program continues by plotting the auto spectra on the Lexidata graphics terminal. (The plotting scales are auto-scaled. The solid lines are the estimated spectra; the dotted lines are standard errors of these estimates.) A menu is displayed on the VT100 terminal, which gives you the option of typing

<return>	to clear the Lexidata screen and continue
P<return>	to produce a copy of the graphics on the printer/plotter
Q<return>	to terminate the the program

The program continues by plotting the estimated spectral coherence between the two data channels and the standard errors of the estimate (dotted lines). The menu is displayed to give you the opportunity of selecting a hard copy of the graphics.

The program continues by displaying a new menu that allows you to change the plotting parameters

P - PLOT POWER SPECTRUM

C - PLOT COHERENCE

F - CHANGE THE FREQUENCY RESOLUTION

Q - QUIT (terminates the program)

These choices are self-explanatory except for choice F, which allows you to replot the power spectra and coherence with a different amount of smoothing applied to the periodogram. Typing F results in the prompt

ENTER FRACTIONAL BANDWIDTH

You may enter RETURN to select the default bandwidth, or type a positive fraction of your choice. The smaller the fractional bandwidth, the greater the frequency resolution of estimated spectra. However, smaller bandwidths also increase the standard errors of the estimated spectra. Therefore, some experimentation is needed to choose a fractional bandwidth that yields the best tradeoff between resolution and stability of the estimated spectra. Typical values of the fractional bandwidth lie in the range of 0.1 to 0.3.

If you type either P to replot the power spectrum, or C to replot the coherence, then the program asks for the type of scales for the X (horizontal) and Y (vertical) axes of the plots. The choices include all possible combinations of linear and logarithmic scales.

The program continues by asking you for the minimum and maximum fr quency limits. You may enter RETURN to select the default value, or type a number of your choice. To invoke autoscaling, type 1 for both the minimum and maximum frequency limits. Autoscaling for the y-axis scales may also be invoked by typing 1 for both minimum and maximum limits.

The program continues by plotting the auto spectra or coherence on the Lexidata graphics terminal. After the plotting is finished, a menu appears on the VT100 monitor to give you the choice of printing a copy of the graphics on the plotter, continuing, or quitting. Typing Q terminates the program. Typing P causes the graphics to be printed on the plotter. Entering RETURN or typing CL clears the screen and continues the program by returning to the menu of plotting options.

3.6 ERROR AND DIAGNOSTIC MESSAGES

The smoothing and spectrum analysis programs have many built-in error traps to flag erroneous data supplied by the user. When you are promped to select an option from a menu, the program checks the validity of any selected option and reprompts you if an invalid option is requested. The program also checks for consistency in the plotting limits and reprompts if some of the values are invalid, such as a zero value to be plotted on a logarithmic scale.

The following message will appear on the screen if the program cannot open and read a data file you specified:

ERROR IN OPENING DATA FILE

The two most likely causes of this error are that you misspelled the file name, or that the file you specified is not compatible with the program you are running. For example, programs GETDATA and FFT can read master data files, while program AR is designed to read data files created by GETDATA, and program PLOTFFT is designed to read data files created by program FFT.

Program AR is intended for analyzing bivariate (two-channel) time series. The program will fail if the time series is degenerate (i.e., one of the channels is identically zero or if both channels contain identical data).

4. USER'S GUIDE TO THE MULTISENSOR SURVEY SIMULATION SOFTWARE

4.1 APPLICATIONS OF THE MULTISENSOR SURVEY SIMULATION SOFTWARE

The multisensor survey simulation software consists of three different programs: MULTISENS, MULTIJOB, and MULTIPLOT. The purpose of these three programs is discussed below.

MULTISENS allows the user to analyze the effects of multisensor surveys of the gravity field on weapon system and gravity estimation errors. The analysis on which the program is based was developed by TASC for the Air Force Geophysics Laboratory and the Defense Mapping Agency and is reported in Ref. 1. The program is ideally suited for the design of multisensor surveys to achieve a desired level of weapon system or gravity recovery accuracy. The program allows the user to consider surveys consisting of any combination of

- Satellite-to-satellite (SST) range-rate data (high-low configuration)
- Gravimetric data (land-based or ship survey)
- Satellite radar altimetry data (up to two missions)
- Airborne gradiometry data.

All parameters which describe the survey geometry and sensor error models can be modified by the user. In addition, the user can define any trajectory profile. The statistics of the unsurveyed (apriori) gravity field can be selected from a

collection of built-in models or can be defined through numerical data tables or through a user-supplied subroutine.

Outputs are printed and/or plotted. They consist of the statistics of postsurvey gravity residuals and the impact miss inflight gravity contribution that would result from compensation using the survey gravity estimates along the trajectory. The statistics of postsurvey gravity residuals are given in the form of root-mean-square (RMS) values, correlation coefficients, spectral densities, and covariances of point values and spatial averages of gravimetric quantities. The output statistics of miss are RMS downrange and crossrange errors as well as their associated CEP.

The programs MULTIJOB and MULTIPLOT serve as tools for the creation of inputs or the analysis of outputs from MULTISENS. MULTIJOB permits the interactive creation of control input files for running MULTISENS, while MULTIPLOT allows the user to examine the output plots produced by MULTISENS, change scales, or obtain hard copies of these plots, all interactively. Recommendations on running the programs MULTISENS, MULTIJOB, and MULTIPLOT are given in sections 4.2, 4.3, and 4.4 respectively.

4.2 RUNNING THE MULTISENS PROGRAM

The MULTISENS program contains three major calculations or phases; a given run may perform all three phases, or any subset of the three. These phases are:

- TRANSFER. This phase of the program reads an input trajectory file and computes the inflight transfer function table, which maps components of the geoid

undulation error spectral density into impact errors for that trajectory. This table is written to a file (TRANSFILE).

- GRAVITY. This phase of the program computes multisensor postsurvey residual gravity error spectral densities and covariances. The program prints and plots the statistics of the residual errors for a set of user-selected gravimetric quantities, and may also write a table of residual geoid undulation spectral densities to a file (GRAVFILE).

- IMPACT. This phase of the program uses the files generated by the other two phases to compute the impact errors associated with the given trajectory arising from the postsurvey residual gravity errors.

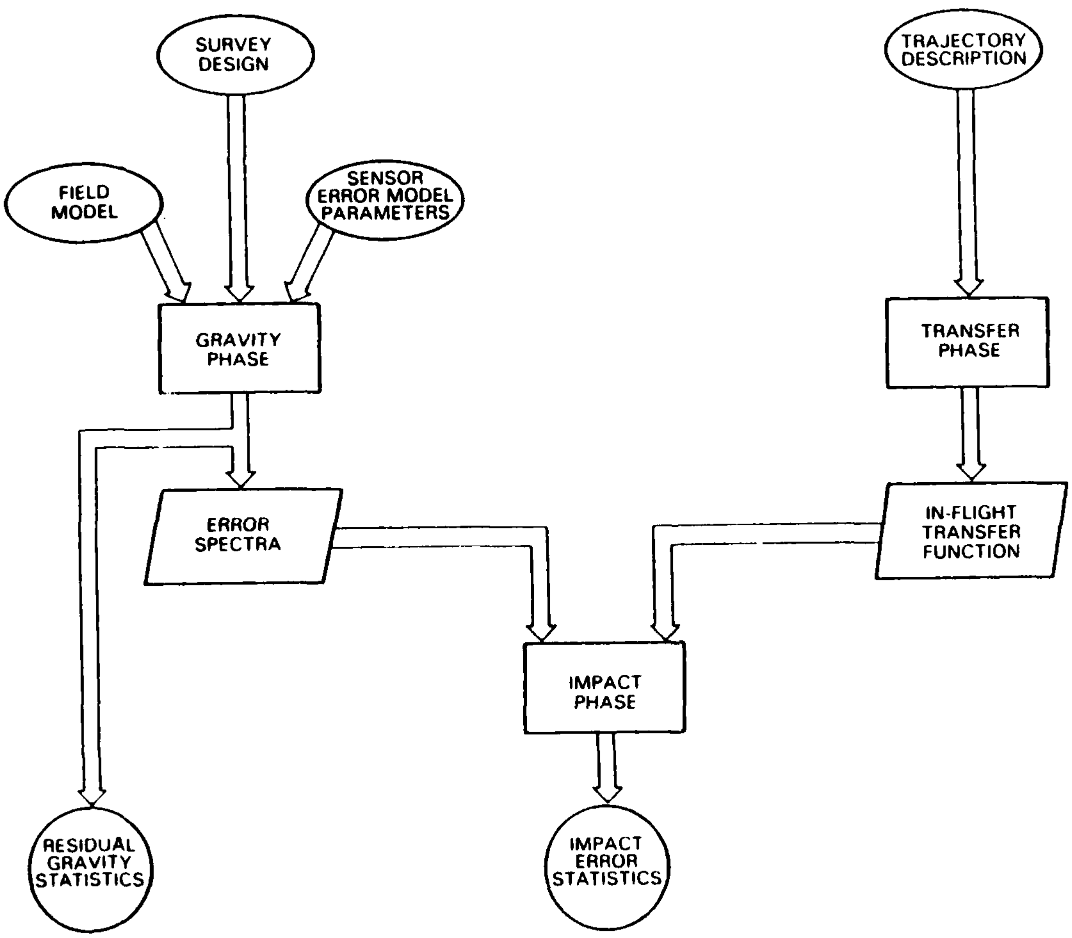


Figure 4.2-1. MULTISENS Functional Macrodiagram

Figure 4.2-1 illustrates the relations among the three phases of the program. The program is designed to enable the user to generate a single transfer function file and then make a series of impact error calculations for that trajectory using various surveys, field models, and/or sensor error model parameter values; alternatively, a single gravity error file may be used in order to compute corresponding impact error statistics for a set of trajectories.

For any MULTISENS run, a single control file is used to specify options for all the phases to be performed. The user may construct this file either by typing the specifications directly into the file using the instructions in this chapter of the User's Guide, or by executing the interactive program MULTIJOB, which prompts the user for run options and generates a control file which may be used as input to MULTISENS. The program can then be run as a batch job using the SUBMIT command by typing

```
SUBMIT FILENAME
```

or it can be run at the terminal using the execute procedure @ command by typing

```
@FILENAME
```

where FILENAME represents the name assigned by the user to the input control file. See Volume 2A of the VAX/VMS Reference Manuals for detailed information on the SUBMIT and @ commands.

MULTISENS reads its control input in the NAMELIST format, which is documented in Appendix B. In this format, the input is read as a series of specifications in the form:

variable-name1 = value1, variable-name2 = value2;

e list being terminated by a semicolon. A semicolon appear-
g in the middle of the file will cause the remaining speci-
cations to be ignored; for this reason character values
ould not include semicolons. Note that the variable names
d alphameric values read from this file are expected to con-
in only upper-case characters and that the order in which
ey are listed is immaterial. A complete list of the variable
mes and default values is provided in Appendix A. In general,
riables whose values are not specified in the input are
signed default values by the program. Exceptions to this
le are the variable TYPE_OF_RUN which indicates what phases
the program are to be executed, and the names of input files,
any are required.

The variable TYPE_OF_RUN, for which a value must be
ecified in the input, is a character string three-vector,
ose elements may be set to any combination of the strings
ANSFER, GRAVITY, and IMPACT. If more than one phase is to
executed, the list of values must be enclosed in parentheses,
in the examples below.

- TYPE_OF_RUN = TRANSFER will cause only the TRANSFER phase to be executed. This run will require an input trajectory file.

- TYPE_OF_RUN = IMPACT will execute only the IMPACT phase. It will require as input both an inflight transfer function file previously generated and saved by the TRANSFER phase, and a residual geoid undulation spectral density file previously generated and saved by the GRAVITY phase.

- TYPE_OF_RUN = (GRAVITY, IMPACT) will execute both the GRAVITY and IMPACT phases; the IMPACT phase will require as input a previously generated inflight transfer-function file, as

well as the output of the GRAVITY phase. The order in which the values are listed does not affect the order in which the phases are executed; the same run could be made by coding TYPE_OF_RUN = (IMPACT, GRAVITY).

- TYPE_OF_RUN = (TRANSFER,GRAVITY,IMPACT) will perform all three phases, requiring only a trajectory file as input.

Any run which includes the execution of the TRANSFER ; requires the specification of an input trajectory file. ie same run specifies IMPACT, the inflight transfer func- table generated during the TRANSFER phase will be used in ;valuation of the statistics of the miss vector.

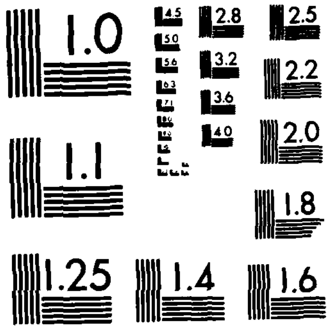
Similarly, if the vector TYPE_OF_RUN includes both ITY and IMPACT, the gravity residuals computed during the ITY phase will be used as input to the impact phase. The :t error statistics will then correspond to the survey itions, gravity model, and sensor error model parameters ified for the GRAVITY phase.

The input control file can be thought of as consist- of up to five blocks as shown in Fig. 4.2-2. The first < contains the single statement describing TYPE_OF_RUN. blocks labeled TRANSFER, GRAVITY, and IMPACT phase controls ain descriptor values which characterize the execution of of these phases. These are discussed in sections 4.2.1, 2 and 4.2.3 respectively.

The SENSITIVITY option control input block of Fig. 2 allows the user to obtain a collection of gravity dual or impact error statistics corresponding to va:ious es of any single input parameter associated with the ITY phase for a fixed trajectory. This option, which is ribed in detail in section 4.2.4, will cause repeated

AD-A156 003 THE WEAPONS SUPPORT SYSTEM USER'S GUIDE(U) ANALYTIC SCIENCES CORP READING MA A R LESCHACK ET AL. 01 JUL 81 TASC-TR-1946-1 AFGL-TR-81-0225 F19628-80-C-0078 2/3

UNCLASSIFIED F/G 8/5 NL


1.0
1.1
1.25
1.4
1.6
1.8
2.0
2.2
2.5
2.8
3.2
3.6
4.0

R-74841

TYPE_OF_RUN SPECIFICATION
TRANSFER PHASE CONTROLS
GRAVITY PHASE CONTROLS
IMPACT PHASE CONTROLS
SENSITIVITY OPTION SPECIFICATIONS

Figure 4.2-2 Input Control File

execution of the GRAVITY phase and of the IMPACT phase if so specified through the SENSITIVITY parameter. Use of this option always requires the inclusion of GRAVITY in the vector TYPE_OF_RUN. If miss statistics are to be evaluated under this option, IMPACT should also be included in TYPE_OF_RUN. If these restrictions are not satisfied by the specification of TYPE_OF_RUN, the setting of SENSITIVITY will cause the program to print warning messages and to modify the values entered for TYPE_OF_RUN in order to construct a self-consistent set of control inputs.

Not all possible settings of the GRAVITY phase control inputs are consistent with the execution of the IMPACT phase in the same run. The only GRAVITY phase outputs compatible with the simultaneous specification of IMPACT in TYPE_OF_RUN are the root-mean-square (RMS) values of the point gravity residuals. This requires that the input GRAVITY parameter MODE be equal to its default value RMS (see section 4.2.2).

4.2.1 Running the TRANSFER Phase

The inflight transfer function describes the relation between high-frequency gravity residuals and impact errors. A complete discussion of the theoretical basis for the definition and construction of the inflight transfer function is given in Ref. 1. For each frequency pair (F1,F2) with F1 and F2 measured in the east and north directions respectively, the inflight transfer function assigns the values of downrange and crossrange miss which are induced on a trajectory by a complex surface potential of the form

$$T(x,y) = \exp\left(i[(F1)x + (F2)y]\right)$$

with $i^2 = -1$; x and y are measured in the east and north directions in such a way that the origin of the coordinate system coincides with the launch point.

The TRANSFER phase of MULTISENS evaluates the inflight transfer function for selected values of the frequency pair (F1,F2) by solving for each pair the differential equations which describe the errors induced by gravity in an inertial navigator. The frequency pairs (F1,F2) form a grid as shown in Fig. 4.2-3. The range of values of F1 is 0, (F0), 2(F0), ... $2^{(NFM1-1)}$(F0) and that of F2 is $-2^{(NFM1-1)}$(F0), ..., 0, ..., $2^{(NFM1-1)}$(F0) with F0 = 1/LAMBDA0 where LAMBDA0 and NFM1 are optional input variables described below. Negative values of F1 are not included in the table since the corresponding values may be derived from the computed entries.

To compute the inflight transfer function, the program needs two input files: a trajectory file describing the path of the vehicle for which the computation is to be made, and the control input which directs the computation.

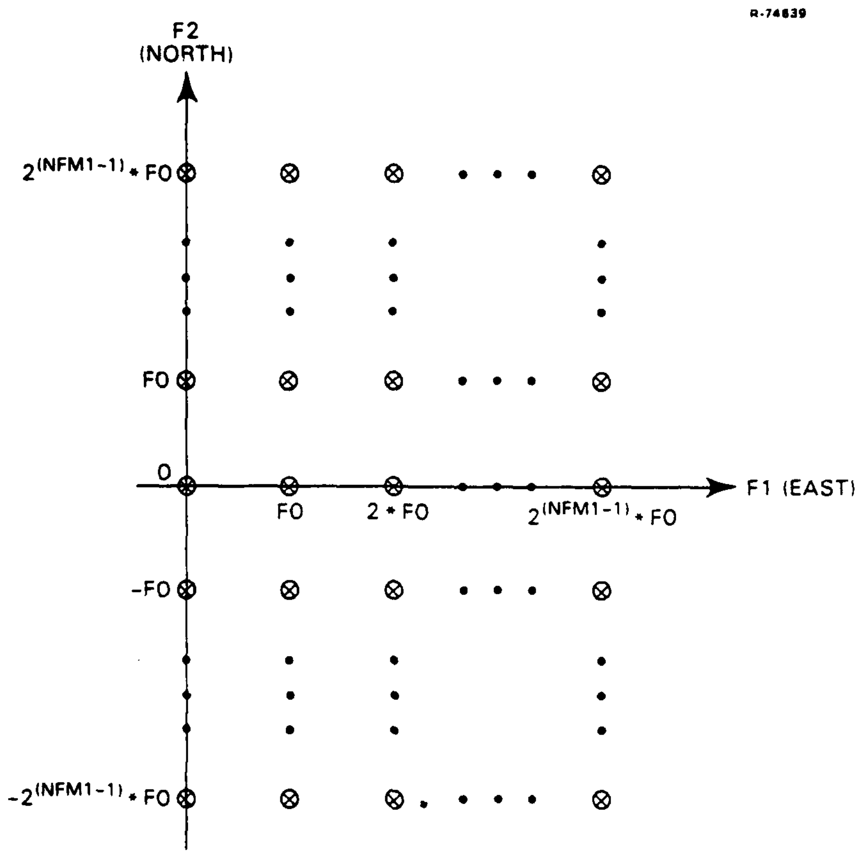


Figure 4.2-3 Frequency Pairs for Transfer Function Table

Trajectory File

The format of the standard input trajectory file is described in section 4.2.5. It should consist of a series of records each containing the time value, position vector, and velocity vector at appropriate intervals in the trajectory.

If the program is required to use a trajectory file in a different format, the user must supply a replacement for the standard MULTISENS subroutine GETTRAJ. Since all access to the trajectory file takes place through this routine, no further program modifications should be necessary. The user's version should return to the program the time, position, and

velocity in the units and coordinate system described in section 4.2.5. For a detailed description of the subroutine and its calling sequence, refer to section 4.2.6.

Control Input

The control input for the inflight transfer function calculation should include:

- A specification for TYPE_OF_RUN that includes the value TRANSFER.
- The filename of the input trajectory file (TRAJFILE).
- The optional filename of the output transfer function (TRANSFILE). If this parameter is omitted, the output will be written to a temporary file, which will be passed to the IMPACT phase (if specified), and deleted at the end of the run. If a filename is specified, the file will be retained after the run is completed.
- Optional specifications for any of the other input parameters (listed below) whose default values are to be overridden.

The following additional parameters are relevant to the TRANSFER calculation.

TDEPLOY This is a time in seconds from launch after which a stepsize of a minimum of ten seconds is sufficient to obtain appropriate results in the propagation of the solution of the differential equations. A reasonable choice for TDEPLOY is the time of deployment for the particular trajectory being analyzed. Unless overriden by the user, its value defaults to 400 sec.

LAMBDA0 Longest non-DC wavelength (in nautical miles) to be used in generation of the transfer-function table. The default is 2400 nm.

NFM1 The number of positive east and north frequency entries to be included in the transfer function table. The default is 10.

CRIT Threshold value for a criterion to determine the stepsize for propagation of the solution. Longer stepsizes are used if the height of the missile is such that the upward continuation attenuation factor for the particular frequency pair being analyzed is less than CRIT. The default value is 0.01.

NWAVE Number of subdivisions of a wavelength to be considered when determining the propagation stepsize. The default value is 12.

DEBUG Logical switch which may be set to generate extra printout from the transfer function calculation. Its default value is .FALSE. (i.e., no debug printout).

MDEBUG Index between zero and NFM1 inclusive. If the DEBUG switch is set, maximum detail printout will be generated for all frequency pairs in which at least one of the frequencies is indexed by a multiple of MDEBUG. The default value is 0.

Printout

Default printout from the TRANSFER phase is limited to the computed inflight transfer function table. Debug printout contains in addition:

- A printout of the contents of an intermediate scratch file, containing a selection of the trajectory points expressed

in local flat-earth coordinates and corresponding rotation matrices relating earth-fixed coordinates to the inertial coordinate system underlying the computation.

- A collection of state transition matrices from launch to selected times along the trajectory.
- Miss partial matrices from launch to impact.
- Tables of sines and cosines used in the computation.
- Tables of propagation time selection, upward continuation attenuation factors, and state solutions (position and velocity errors) as functions of time for all frequency pairs (F1,F2) in which at least one of the frequencies is indexed by a multiple of MDEBUG.

Output File

The computed inflight transfer function is written to an output file whose name is specified as the TRANSFILE parameter in the control input. If the TRANSFILE parameter is missing from the control input, a temporary file is created, which is deleted at the end of the run. The format of this file is described in section 4.2.5.

4.2.2 Running the GRAVITY phase

The GRAVITY phase computes the statistics of multisensor postsurvey residual gravity errors. The theoretical basis for the evaluation of the postsurvey residuals is given in Ref. 1. The two-dimensional power spectral density of the residuals in the undulation of the geoid, S, is given at each frequency pair (W1,W2) by

S = 1 / (A(1) + A(2) + ... + A(N) + 1/S0)

where S0 is the power spectral density (PSD) of the unsurveyed (apriori) geoid undulation, A(K) is a spectral error term computed on the basis of the geometry and sensor models for the K-th survey, and N is the total number of independent surveys to be considered. The statistics of gravimetric quantities other than the undulation of the geoid are obtained through the use of flat-earth frequency-domain relations.

There are two main outputs of the calculation. They are:

- The statistics of the residual errors in up to ten gravimetric quantities. The user may select any subset of the ten available quantities. Depending on the output options selected (see the section <u>Selection of Outputs</u> below), these statistics may include one-dimensional spectral densities, auto- and crosscovariances, or standard deviations and correlation coefficients.

- A file containing the two-dimensional power spectral density of the postsurvey residual undulation of the geoid. This file, which may be used for subsequent IMPACT runs, can be obtained as an output file only when the parameter MODE satisfies certain restrictions (see the section <u>Selection of Outputs</u> below).

Input Files

The only input file necessarily required for this phase is the control file, in which the TYPE_OF_RUN specification should include the value GRAVITY. If the NUMERIC gravity model is to be used (see below), additional files containing tables of model geoid undulation spectral densities are also required. These files are described in section 4.2.5.

The run options available in the GRAVITY phase are decribed below. Parameters which may be used to select and/or modify outputs, models for the gravity field, survey geometry, and sensor error models are discussed in detail.

Selection of Gravimetric Quantities

The gravimetric quantities whose error statistics are to be computed and printed may be specified in the variable GRAVERR, which is a vector of type CHARACTER and can accept up to ten values. The available quantities are:

UND	= Undulation of the geoid
NORTHVD	= North deflection of the vertical
EASTVD	= East deflection of the vertical
ANOMALY	= Gravity anomaly
NNGRAD	= North-North gravity gradient
EEGRAD	= East-East gravity gradient
UUGRAD	= Up-Up gravity gradient
NEGRAD	= North-East gravity gradient
NUGRAD	= North-Up gravity gradient
EUGRAD	= East-Up gravity gradient

Any subset of the above strings may be specified in any order. For example, the entry GRAVERR=(ANOMALY,NORTHVD,EASTVD) will compute error statistics for the three quantities in the above list in the order specified; i.e., the power spectral density and covariance matrices will be printed with the (1,1) element representing the gravity anomaly PSD and/or covariance, the (1,2) element would correspond to the cross-spectral density and/or crosscovariance between the gravity anomaly and the north deflection of the vertical, etc. The default value is

GRAVERR=UND, which will compute only the geoid undulation statistics. The GRAVERR specification has no effect on the generation of the output file containing the geoid undulation spectral density which is passed to the IMPACT calculation.

Selection of Outputs

The GRAVITY segment of the program permits a variety of quantities to be computed, depending upon the options selected by the user. These options are entered via the scalar character string MODE.

The variable MODE will accept any one of four values: PSD, COV, RMS, and AREAMEAN. The default setting is RMS.

A setting of MODE=PSD will cause the program to scan through the two-dimensional frequency domain grid shown in Fig. 4.2-4. The extent of the grid and the number of gridpoints are controlled by the parameters SAMPLEA, SAMPLEC, M1, and M2. The input parameters SAMPLEA and SAMPLEC control the highest along track and cross track frequencies, respectively, which are covered by the grid. (See Fig. 4.2-4). Their values are given in nautical miles and both default to 1.0 when MODE=PSD The parameters M1 and M2 control the number of grid points in the along track and cross track directions respectively. Their values default to 100 when MODE=PSD. The frequency range scanned and the density of the grid points in each direction may be controlled by overriding the default values of SAMPLEA, SAMPLEC, M1, and M2. The resultant array of residual geoid undulation spectral densities is written to the file GRAVFILE. If a filename has been assigned to GRAVFILE in the control input, the file is kept after completion of the run, otherwise the file is deleted.

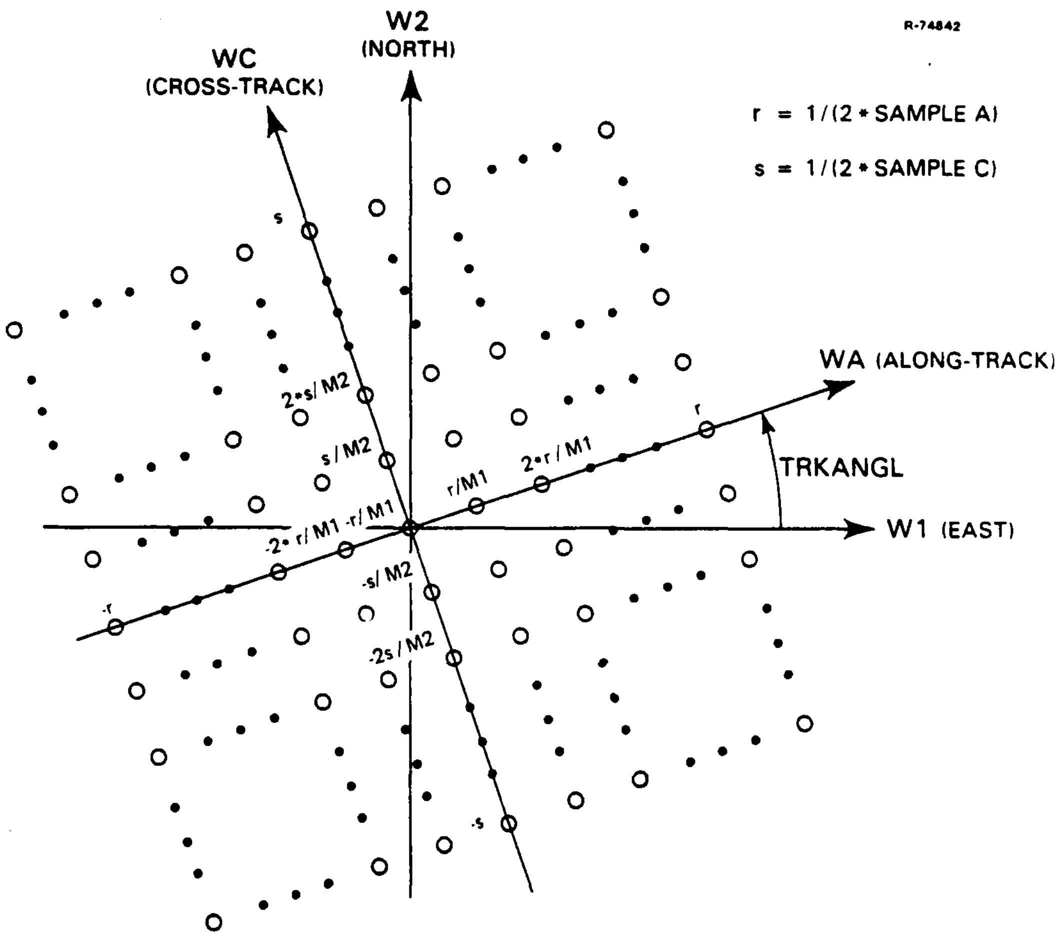


Figure 4.2-4 Frequency Domain Grid for Evaluation of Residuals

In addition, under MODE=PSD the program will use the residual geoid undulation spectral density to compute, at each grid point, the two-dimensional spectral density matrix of the gravimetric quantities specified in the vector GRAVERR. These matrices are integrated in the crosstrack direction to produce a one-dimensional spectral density matrix in the direction specified by the parameter TRKANGL (along track direction measured counter clockwise from east). The resulting densities

are then printed and written onto the file PLOTFILE if a filename has been specified for this parameter in the control input.

A setting of MODE=COV will do all the above, but in addition will compute approximations to the auto- and cross-covariances of the gravity residuals using a mixed-radix FFT algorithm on the one-dimensional spectral densities to compute their Fourier transforms. The zero-shift component of this array is extracted and printed as the overall covariance matrix of the gravimetric quantities selected. The covariance functions generated by the above procedure are also printed. The files GRAVFILE and PLOTFILE generated by this option are identical to those generated for MODE=PSD, except that PLOTFILE contains the covariance and crosscovariance functions in addition to the spectral densities. Saving of either the file GRAVFILE or the file PLOTFILE is possible only with the settings MODE=PSD and MODE=COV.

A setting of MODE=RMS is used to compute the root-mean-square values, standard deviations, and correlation coefficients of point values of the postsurvey residuals in all gravimetric quantities listed in GRAVERR. The implementation of the MODE=RMS option is dictated by the fact that the grid used for the computations is uniformly spaced in the frequency domain. To obtain appropriate coverage of the frequency plane with a single grid such as that of Fig. 4.2-4 would require extremely large values for the parameters M1 and M2. Since storage requirements and computation time grow as the product of M1 and M2, use of a single grid is not a practical alternative.

To avoid this problem, the RMS mode of operation effectively divides the frequency domain into three concentric square regions as shown in Fig. 4.2-5, of which the innermost region is integrated on a finely spaced grid, the second on a coarser

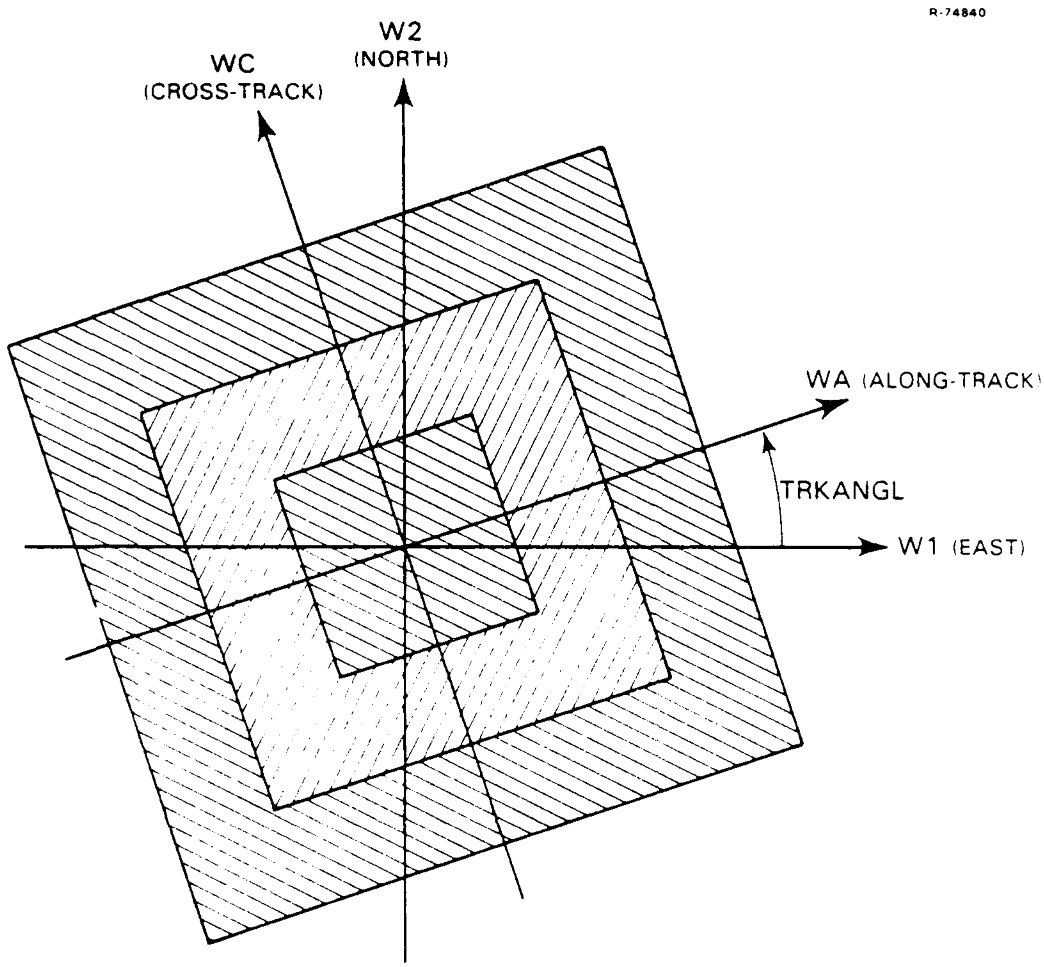


Figure 4.2-5. Frequency Domain Grids for Multiple Scans

is integrated on a finely spaced grid, the second on a coarser grid, and the outermost on the most widely spaced grid. This is accomplished by performing a total of five passes through the procedure described above for the MODE=COV option. A single pass is made through the central low-frequency region using the fine grid spacing, and the computed gravimetric covariance contribution from integrating over this region is saved. The contribution to the covariance obtained from the intermediate-frequency region is computed by performing two more passes with the intermediate grid spacing, one including both the intermediate and low-frequency regions, and the second including only the low-frequency region; the difference in the gravimetric covariances obtained from these two integrations provides

SAMPALTP Time in seconds between successive altimeter measurements. This value must be positive, and defaults to 0.1.

CAP White noise level in meters squared per measurement of the altimeter. This value must be positive, and defaults to 0.01.

ALTVARP Variance of the altimeter bias in meters squared. This value must be positive, and defaults to 0.25.

ALAMP Alongtrack inverse standard deviation of the bias model, in inverse meters. This value must be positive, and defaults to 10^{-6}.

In addition to the above parameters, the oceanographic effects for both altimeter surveys can be controlled through the parameters SIGMAC2 and BETAC. These parameters are described below under the next heading.

Gravimeter Survey Parameters

The gravimeter surveys are modeled as patterns of uniformly spaced east-west tracks, with measurements spaced at equal intervals along each track. The input parameters for either survey are listed below; they are real scalars, except where otherwise indicated.

SE The data spacing in nautical miles in the east direction (i.e., along the tracks of the survey ship in the case of GRAVSHIP). This value must be positive, and defaults to 0.25 for the GRAVSHIP survey, and 81 (150 km) for the GRAVLAND survey.

SN The data spacing in nautical miles in the north direction (i.e., the distance between the tracks of

TRKSP — Distance in nautical miles between the equatorial crossings of the ground tracks, for both ascending and descending nodes of the satellite. This value must be positive, and defaults to 30.

SAMPALT — Time in seconds between successive altimeter measurements. This value must be positive, and defaults to 0.1.

CA — White-noise level in meters squared per measurement of the altimeter. This value must be positive, and defaults to 0.36.

ALTVAR — Variance of the altimeter bias in meters squared. This value must be positive, and defaults to 0.25.

ALAM — Alongtrack inverse standard deviation of the bias model, in inverse meters. This value must be positive, and defaults to 10^{-6}.

Parameters describing the ALT2 survey are all real scalars. They are:

ORBINCP — Inclination in degrees of the satellite orbit. This parameter must lie between the absolute value of parameter EL (the mean latitude of the region under analysis), and 180 minus the absolute value of EL; it defaults to a value of 108.

HEITALTP — Altitude in meters of the satellite above the surface. This parameter must be positive, and defaults to a value of 8×10^5 (800 km).

TRKSPP — Distance in nautical miles between the equatorial crossings of the ground tracks, for both ascending and descending nodes of the satellite. This value must be positive, and defaults to 80.

and with carouselling, respectively. The survey type GOAL assumes a one-Eotvos gradiometer. GRAD indicates a nonstandard gradiometer whose red and white noise levels may be defined by the user. Error models for the standard gradiometers are described in detail in Ref. 4.

A detailed description of the input parameters associated with each of the possible surveys is given below. These parameters describe both the survey geometry and the sensor error models.

Satellite Radar Altimeter Survey Parameters

Each satellite altimeter survey is modeled as a uniform pattern of ascending and descending subsatellite ground tracks, with measurements spaced uniformly along each track. Values of the input parameters describing this pattern and the instrument's performance may be included in the control file. Variable values not specified default to those applicable to the GEOS-3 and SEASAT-1 missions for the ALT1 and ALT2 surveys respectively.

Parameters describing the ALT1 survey are all real scalars. They are:

ORBINC	Inclination in degrees of the satellite orbit. This parameter must lie between the absolute value of parameter EL (the mean latitude of the region under analysis), and 180 minus the absolute value of EL; it defaults to a value of 115.
HEITALT	Altitude in meters of the satellite above the surface. This parameter must be positive, and defaults to a value of 8×10^5 (800 km).

accept up to five values; the user may select no more than one survey descriptor from each of the following five groups:

- For the first satellite radar altimeter survey: ALT1
- For the second satellite radar altimeter survey: ALT2
- For gravimeter surveys: GRAVSHIP or GRAVLAND
- For satellite-to-satellite Doppler surveys: SST
- For airborne gradiometer surveys: DRAPER, DRAPERC, BELL, BELLC, GOAL, or GRAD.

Thus, for example, the specification

SURVEY = (ALT1, ALT2, DRAPERC)

is correct, but

SURVEY = (ALT1, DRAPERC, GOAL)

is not correct because two survey types from the last group have been included.

If the SURVEY specification is omitted from the input, the statistics of the unsurveyed quantities listed in GRAVERR will be computed.

The descriptors GRAVSHIP and GRAVLAND indicate ship-based and land-based gravimeter surveys respectively. Of the gradiometer descriptors, BELL and BELLC indicate the Bell gradiometer without and with carouselling, respectively; DRAPER and DRAPERC indicate the Draper Laboratory gradiometer without

element is specified for USERFILE the program will use the same filename for the other two components. If two members are specified, they are assigned to the innermost and intermediate region computations in the order in which they are listed. In this case the second filename is also used for the outer region. If the region being analyzed is larger than the frequency-domain region covered by the contents of the file being used, the program will zero the model spectral density outside the range of values contained in the file. Therefore special care is recommended in ensuring that the frequency ranges of the file(s) are compatible with the ranges corresponding to the parameters in Table 4.2-1.

The units of the tabulated values of the two-dimensional power spectral density of the undulation of the geoid are assumed to be $m^2/(cycle/m)^2$. If different units are used, the variable USDFACT may be assigned a value in the control input to convert the spectral density values into the appropriate units. All tabulated values are multiplied by the variable USDFACT which defaults to 1.

If the MODEL=USER option is selected, the user will be expected to provide and link into the program a FORTRAN subroutine USRFUNC which returns the two-dimensional geoid undulation spectral density as a function of east and north frequency values. Details of the USRFUNC calling sequence are provided in section 4.2.6. The version of USRFUNC supplied with the program simply generates an error message and terminates execution.

Survey Selection

The surveys to be considered in evaluating the post-survey gravity error statistics are specified in the input parameter SURVEY. This is a character string vector which can

BASELINE model and 2.856914×10^{-6} for the ACTIVE model. The corresponding characteristic distances are 200 nm for the BASELINE model and 189 nm for the ACTIVE model.

All these parameters are declared REAL. The values of SIGMN12 and SIGMN22 must be greater than or equal to zero while those of BETAN1 and BETAN2 must be strictly positive.

The AWN (Attenuated White Noise) model contains ten different parameters. It corresponds to a fit to a global data set (Ref. 3). The model parameters are fixed in the program and cannot be modified by the user. The model corresponds to a smooth gravity field rick in long wavelength energy content but having little energy at high frequencies.

The NUMERIC model requires that the user supply one, two, or three input files. File names are entered via the vector character parameter USERFILE in a statement such as

USERFILE = (filenamea, filenameb)

Each file must contain geoid undulation spectral density values tabulated on a unfirom grid of east and north frequencies. The program will use a bilinear interpolator on the logarithm of the grid point values to determine spectral density values between grid points. The format for these files is described in section 4.2.5. If MODE=PSD or MODE=COV, only the first filename listed for USERFILE is used; other entries in userfile are ignored. Multiple files will be used only for the MODE=RMS and MODE=AREAMEAN options previously discussed. In this case the program will use the first member of the USERFILE in evaluating contributions from the innermost région of Fig. 4.2-5. The second member is used for the intermediate grid and the third for the high frequency grid. If only one

third-order isotropic Markov terms (Ref. 2). Each Markov term is determined by two parameters: variance and characteristic distance. Therefore, the BASELINE and ACTIVE models each contain four parameters, the differences between the two models being reflected in the values of these parameters. The ACTIVE model, as its name indicates, corresponds to an active gravity field. Its parameters were obtained by fitting to data in the Bonin Trench. The BASELINE model represents a somewhat milder gravity field. Its parameters were determined from data in the North Atlantic. The parameters of these models are:

SIGMN12 — Variance (in units of meters squared) of the undulation of the geoid associated with the first third-order Markov term. Defaults are 2.666438375 for the BASELINE model and 9.367056530 for the ACTIVE model. When combined with the values of BETAN1 given below, these numbers yield deflections of the vertical with standard deviations of 7.0 sec for the BASELINE model and 16.4 sec for the ACTIVE model.

SIGMN22 — Variance (in units of meters squared) of the undulation of the geoid associated with the second third-order Markov term. Defaults are 87.06737555 for the BASELINE model and 111.9648140 for the ACTIVE model. When combined with the values of BETAN2 given below, these numbers yield deflections of the vertical with standard deviations of 3.0 sec for the BASELINE model and 3.6 sec for the ACTIVE model.

BETAN1 — Inverse characteristic distance (in units of inverse meters) of the first third-order Markov term. Defaults are 0.000036 for the BASELINE model and 0.0000449964 for the ACTIVE model. The corresponding characteristic distances are 15 nautical miles for the BASELINE model and 12 nm for the ACTIVE model.

BETAN2 — Inverse characteristic distance (in units of inverse meters) of the second third-order Markov term. Defaults are 2.7×10^{-6} for the

using MULTIPLOT. In this case the vector PLOT must specify at least one admissible option compatible with the MODE and SENSITIVITY settings.

If MULTISENS is run at a terminal and plotting options are desired, PLOT must be given at least one admissible value. The user will then be prompted for all possible options during execution. Note that no plots will be displayed in this case if a filename has been assigned to PLOTFILE.

Gravity Model Selection

MULTISENS allows for several possibilities in the description of the unsurveyed gravity field. Three analytic gravity models are included. There is also the option of using a numeric model which requires external files supplied by the user. In addition, there is a stub for attaching a FORTRAN subroutine written by the user defining the gravity field model. A stationary gravity field model is specified in MULTISENS by the two-dimensional spectral density of the undulation of the geoid. Spectral densities, covariances, and crosscovariances of all other gravimetric quantities are obtained from this two-dimensional density through the use of algebraic relations in the frequency domain.

The gravity model to be used is determined by the user in the control input file by setting the character-string scalar parameter MODEL equal to one of the five values: BASELINE, ACTIVE, AWN, NUMERIC, and USER. The default setting is BASELINE.

Both BASELINE and ACTIVE field models are members of the same class of models. They are sums of two independent

PLOT=COR	This produces plots of alongtrack crosscorrelations of the residual errors between pairs of the selected gravimetric quantities. The scales default to linear in both variables.
PLOT=3-D	This produces a perspective plot of the spectral density of geoid undulation residuals on a two-dimensional east and north frequency domain. In the east direction, this domain ranges from zero to the maximum alongtrack frequency (as computed from SAMPLEA) and in the north direction it ranges from minus to plus the maximum crosstrack frequency (as computed from SAMPLEC). The number of grid points in the east and north directions may be specified as M1PLOT and M2PLOT respectively; the grid points are then indexed to range from zero to M1PLOT in the east direction, and from -M2PLOT to M2PLOT in the north direction. Both M1PLOT and M2PLOT default to 50. The scales are linear on the frequency axes and logarithmic on the vertical axis. The scales cannot be changed, but the viewing angle may be varied interactively.
PLOT=CONTOUR	This produces a contour equivalent of the 3-D plot; it may not be altered interactively. The parameters M1PLOT and M2PLOT also control the number of grid points generated for this plot.

If the program is run in batch form and no PLOTFILE filename is included in the control input, only those plots indicated by the vector PLOT will be printed using the default scales. When PLOTFILE is specified in the control input, the file will contain information to generate all possible plots available under the MODE or SENSITIVITY options in effect,

values from PSD, 3-D, and CONTOUR. When MODE=COV, PLOT can contain any number of values from PSD, COH, COV, COR, 3-D, and CONTOUR. The various options are described below.

PLOT=SENS	This option generates plots of the dependent variables against the independent variable of the sensitivity run. Sensitivity plots are spline-interpolated, using the IMSL procedure ICSICU. For SENSITIVITY=IMPACT or for SENSITIVITY=RMS with IMPACT included in TYPE_OF_RUN, the dependent variables plotted are the standard deviations of the selected gravimetric quantities, in addition to the downrange and crossrange standard deviations and the CEP. For SENSITIVITY=AREAMEAN, or for SENSITIVITY=RMS with no IMPACT phase, only the gravimetric standard deviations are plotted. Default scales, unless interactively modified, are linear in both variables with the units given on the plots.
PLOT=PSD	This produces plots of the alongtrack spectral densities of the residual errors in each of the selected gravimetric quantities. The scales default to logarithmic in both directions (zero frequency is not shown).
PLOT=COH	This produces plots of the alongtrack coherence of residual errors between pairs of the selected gravimetric quantities. The default scale is linear for the ordinates and logarithmic for the abscissas.
PLOT=COV	This produces plots of the alongtrack autocovariance functions of the residual errors in the selected gravimetric quantities. The scales default to linear in both variables.

indicate the specific plots to be generated. The disposition of MULTISENS plots is controlled by the input parameter PLOTFILE, a character-string scalar which accepts an output filename as a value.

If PLOTFILE is set to a filename, no graphic plots are generated in the MULTISENS run; instead, the plot information is written to the filename specified, and the interactive program MULTIPLOT can be used subsequently to view and print the plots, as well as modify scales. The use of MULTIPLOT is documented separately in section 4.4.

If no value is entered for PLOTFILE, the graphics are generated in the MULTISENS run. In this case, for a batch run the plots are directed to the printer. If the program is run at the terminal the plots will be constructed on the graphics display screen, and the user will be prompted for optional modifications to the plot and hardcopy generation. If the graphics terminal is allocated to another user when the program is run, the screen display will not appear; however, the program will execute as if the plot was being displayed. Valid responses to the prompts (including hardcopy generation) will be executed, but the user will be unable to see the graphics prior to printing.

If the PLOT parameter is not set, or if TYPE_OF_RUN does not include the GRAVITY phase, no plots will be generated. PLOT is a character-string vector which can accept subsets of seven values; these values are: PSD, COH, COV, COR, 3-D, CONTOUR, and SENS. Not all plot options can be selected in the same run. The limitations are as follows. For a sensitivity run only PLOT=SENS may be used. For runs not using the sensitivity option the MODE parameter must be set to either PSD or COV. When MODE=PSD the vector PLOT may be assigned any

A setting of MODE=AREAMEAN is used to compute statistics of spatial averages of the selected gravimetric quantities. In this mode, the program will not generate the file GRAVFILE for the calculation of impact errors, since in the evaluation of spatial averages, geoid undulation spectral density values at different frequencies are not weighted in the same manner. The spatial averages are assumed to be taken over square regions whose size is determined by the input parameter AREASIZE (the value of which is given in $\widehat{\text{min}}$). In other respects, the computation resembles a MODE=RMS run; the spectral density of the spatial average residual errors is integrated over each of the three frequency regions, and the covariances, standard deviations, and correlations of the residuals in the spatially averaged gravimetric quantities are printed. The inclusion of gravity gradients among the quantities statistics are to be computed does not double the size of the frequency domain scanned in an AREAMEAN run, as it does in a RMS run.

If the SENSITIVITY option is not used and AREASIZE is not specified in the control input file, MODE=AREAMEAN will cause the program to evaluate successively the statistics of the 5 $\widehat{\text{min}}$, 15 $\widehat{\text{min}}$, 1 deg and 5 deg means of all gravimetric quantities listed in the vector GRAVERR. If the SENSITIVITY option is not used but a value has been assigned to AREASIZE, only the statistics of AREASIZE $\widehat{\text{min}}$ means will be produced. When the SENSITIVITY option is used, SENSITIVITY must equal AREAMEAN for the computation of the residuals in spatially averaged quantities. In this case, AREASIZE defaults to the single value of 60 $\widehat{\text{min}}$ (1 deg means) unless a different value is entered for it in the control input file.

Plotting Options

Plot generation by MULTISENS is requested by setting the parameter PLOT, a character-string vector whose values

TABLE 4.2-1

PREDETERMINED VALUE OF INTEGRATION CONTROL PARAMETERS FOR THE RMS AND AREAMEAN MODES

PASS DESCRIPTION	SAMPLEA, SAMPLEC	M1,M2
Low frequency finest grid	94.5000	50
Intermediate frequency intermediate grid	8.4208	50
Low frequency intermediate grid	105.259	4
High frequency coarsest grid (for MODE=RMS runs including gradients)	0.3752	100
High frequency coarsest grid (for all other cases)	0.7504	50
Intermediate frequency coarsest grid	9.3796	4

The IMPACT calculation will be performed after each of the five scans, and the five computed impact covariance contributions are combined in the same way as the gravimetric covariances to yield the total impact covariance. Because each scan after the first rewinds and rewrites the file GRAVFILE, the option to save GRAVFILE for subsequent IMPACT runs is not available in the RMS mode, since the file remaining at the end of the run would correspond to the last scan only. Recommendations on setting GRAVITY phase input parameters for the purpose of saving GRAVFILE for future use in IMPACT runs are given in section 4.2.3.

the contribution of the intermediate-frequency region only. A similar two-pass procedure is used to obtain the contribution from the high-frequency region only, these scans being performed using the coarsest grid spacing. The sum of the contributions from all three regions to the gravimetric covariance provides total covariances corresponding to integration over the entire relevant frequency range.

The frequency range and grid spacing of an individual pass are normally controlled by the input parameters SAMPLEA, SAMPLEC, M1, and M2. For the series of concentric scans described above, values of these quantities have been precomputed and are fixed in the program. The RMS option will automatically use these predetermined constants for each pass. Attempts to override these values in the control file will be ignored. The values of SAMPLEA, SAMPLEC, M1, and M2 associated with each of the five scans are listed in Table 4.2-1. As indicated in the table, the range of frequencies covered by the high-frequency scan is doubled for cases in which the selected gravimetric quantities include gravity gradients; this is because the gradients are one order higher derivatives of the potential than the other quantities, and so may produce non-negligible contributions at higher frequencies. Printed output from the RMS option consists of covariances, standard deviations, and correlation coefficients of the residuals in the gravimetric quantities listed in GRAVERR.

Note that if the TYPE_OF_RUN specification also includes the IMPACT phase, the parameter MODE must be set to RMS. Any other specification for MODE is overriden by the program and a warning message is printed.

the survey ship in the case of GRAVSHIP). This value must be positive, and defaults to 6 for the GRAVSHIP survey, and to 81 (150 km) for the GRAVLAND survey.

CG	White noise measurement error level, in $(m/sec^2)^2$. This value must be positive, and defaults to 10^{-12}(0.1 mgal rms) for the GRAVSHIP survey and 9×10^{-10} (3.0 mgal rms) for the GRAVLAND survey. This is the only measurement error term used in the GRAVLAND survey and should include errors of reduction to the geoid. Measurement error sources for the GRAVSHIP survey consist of instrument noise (controlled by CG), Eotvos correction errors (controlled by QQ and EOTVOSW) and ocean-current induced sea surface height (controlled by SIGMAC2 and BETAC).
QQ	Root-mean-square uncorrelated error in the Eotvos correction, in mgal per measurement. This value must be positive, and defaults to 0.75.
EOTVOSW	A logical switch to determine whether the Eotvos correction error terms are to be included in the calculation. It defaults to .TRUE. for the GRAVSHIP survey, and .FALSE. for GRAVLAND. The user may override this default for GRAVSHIP only.
FULFUNC	An integer switch to determine whether alongtrack (east) aliasing terms are to be included in the calculation. A value of zero omits the terms, a value other than zero includes them. The default is zero for GRAVSHIP and 1 for GRAVLAND.
GRAVXTENT	The upper bound in nautical miles for wavelengths to be included in modeling the ship survey. If the

value is zero or negative, there will be no upper bound on the wavelengths included. This parameter defaults to 150 for the GRAVSHIP survey, and -10 for GRAVLAND.

SIGMAC2 — The variance in meters squared of the uncorrected ocean current induced sea-surface height. It must be nonnegative and defaults to 0.36 (60 cm rms).

BETAC — The inverse characteristic distance in inverse meters for the third-order Markov model associated with the ocean current induced sea-surface height. It must be positive and defaults to 2.3809524×10^{-5} (the characteristic distance is 42 km and the correlation distance is approximately 122 km).

Satellite-to-Satellite Doppler Parameters

The satellite-to-satellite Doppler coverage is modeled as a uniform pattern of ascending and descending ground tracks of the lower satellite, similar to the pattern of altimeter coverage. The range-rate measurements take place between this lower satellite and the geostationary relay satellite. The input parameters modifying the SST survey are:

LOORBINC — Orbital inclination of the low satellite (deg). This parameter must lie between the absolute value of parameter EL (the mean latitude of the region under analysis), and 180 minus the absolute value of EL; it defaults to a value of 86.

LOHEIGHT — Altitude in kilometers of low satellite above ground. This parameter must be positive, and defaults to 150.

SSTTRKSP — Distance in nautical miles between the equatorial crossings of the lower satellite's ground tracks. This parameter must be positive, and defaults to 30.

SSTSP — Time between successive Doppler measurements (sec). This parameter must be positive, and defaults to 10.

HIHEIGHT — Altitude in meters of the geostationary satellite above ground. This parameter must be positive, and defaults to 3.5786×10^7.

SSTLONG — Longitude of estimation region relative to the position of the geostationary satellite (deg). The default is zero.

SSTNOISE — Root-mean-square white noise level of the range-rate measurements, in micrometers per second. This parameter must be positive, and defaults to 100.

SSTFUNC — Integer switch to indicate whether full aliasing is to be included in the computation. A value of zero permits a simplified aliasing calculation, while all other values generate the more complex model. The default is 0.

Airborne Gradiometer Survey Parameters

The various airborne gradiometer surveys are modeled as a uniform pattern of east-west parallel tracks, with measurements at equal intervals along them. The gradiometer survey parameters are:

GRADSP — Distance between tracks, in kilometers. This parameter must be positive, and defaults to 10.

SAMPINT	Time in seconds between samples. This parameter must be positive, and defaults to 10.
HEIGHT	Height of aircraft above ground, in meters. This parameter must be positive, and defaults to 6096 (20,000 ft).
SPEED	Speed of aircraft, in km/hour. This parameter must be positive, and defaults to 555.6 (300 knots).
GRADXTENT	The longest wavelength in nautical miles which is to be included in modeling the survey. If the value is zero or negative, there no upper bound on the wavelengths included. This parameter defaults to 1500.
GRADFUNC	An integer switch to indicate whether full aliasing is to be included in the computation. If GRADFUNC = 0 a simplified aliasing computation is used. All other values use the more complex model. The default is 1.
SWGRAD(3)	A three-vector of logical switches, which activate the Z, Y, and X components of the gradiometer triad respectively. If one of the gradiometer surveys is selected via the SURVEY parameter, but none of these switches is explicitly set to .TRUE. by the control stream, the program will set all three to .TRUE. and all gradiometer components will be active. However if the user explicitly sets to .TRUE. a subset of these switches, only that subset will be active.
GRADROT	A logical switch which controls whether the umbrella geometry is to be used for the instrument. For the Draper gradiometer surveys, this parameter defaults to .TRUE. and cannot be overridden; for all other gradiometer survey types, it defaults to .FALSE. but can be overridden by the user.

TABLE 4.2-2

GRADIOMETER SURVEY PARAMETERS

SURVEY	RDNSE	WHTNSE
DRAPER	2×10^{-7} (all components)	2.3 (all components)
BELL	7.7×10^{-6} (all components	650 (first two components), 300 (last four components)
DRAPERC	0 (all components)	2.3 (all components)
BELLC	0 (all components)	650 (first two components), 300 (last four components)
GOAL	0 (all components)	10 (all components)

RDNSE(6),WHTNSE(6) Vectors of coefficients for red and white noise levels (see Ref. 4). The units of these coefficients are E^2Hz for RDNSE and E^2/Hz for WHTNSE, where E is the Eotvos unit. For all survey types except GRAD, these vectors are set by the program and cannot be overridden. For the GRAD survey, they default to 10^{30} and should be overridden by the user. The override values must be positive. Settings used for the other surveys are shown in Table 4.2-2.

Note that the BELL gradiometer triad white noise levels include linear and rotational vibration sensitivity effects aside from the instrument's self-noise. Vibrationally induced errors are taken to be the same at all aircraft altitudes. None of the other built-in gradiometer error models contains these effects.

Additional optional input parameters for the GRAVITY phase:

TRKANGL Angle in deg with respect to east (see Fig. 4.2-4) for definition of the alongtrack direction in the computation of the one-dimensional

spectral densities and covariances. Its value defaults to zero. When used in an IMPACT run with a value different from zero, the results approximate the miss statistics for a trajectory whose bearing from north is that of the original trajectory used to create the inflight transfer function minus the value of TRKANGL. The approximation relates only to the effects of the rotation of the earth.

ALTITUDE	Postsurvey gravimetric error statistics may be computed at any height above the earth's surface. The ALTITUDE parameter is used to select the desired height. ALTITUDE is measured in meters above the surface; it must be nonnegative, and defaults to zero. Note that the undulation error spectrum written to the GRAVFILE file for use in the IMPACT calculation is always computed at the surface, so that the ALTITUDE parameter will have no effect on the impact errors. For that reason it cannot be used as the independent variable in an impact sensitivity run.
EL	Mean latitude of the region in which the gravity field is being estimated. This parameter is expressed in deg, and defaults to zero. Its absolute value must be such that the region lies within the latitudes covered by the applicable satellites, if any of the ALT1, ALT2, or SST surveys are selected. (See documentation on parameters ORBINC, ORBINCP, and SSTORBINC.)
GRIDSAVE	For runs in modes PSD or COV, it is possible to save computer time (losing some accuracy in the computations) by using a more widely spaced crosstrack grid spacing for large alongtrack frequency values, since the spectral density function is expected to vary slowly at these frequencies. The parameter GRIDSAVE indicates the fraction of alongtrack

frequency values which are to be integrated using the grid spacing indicated by the M2 and SAMPLEC settings; the remaining alongtrack frequency values (at the high-frequency end of the spectrum) will be integrated using a grid spacing four times wider. For example, given the (partial) input:

MODE=PSD, M1=100, M2=100, GRIDSAVE=0.6

the program will integrate each of the first 61 values of the alongtrack spectral density using 100 crosstrack grid points; the remaining 40 alongtrack frequency values will be integrated using 25 crosstrack gridpoints spaced four times further apart. The parameter GRIDSAVE is real, and must lie between 0.5 and 1 inclusive. It defaults to 1, and may be overridden only for MODE=PSD or COV. Note that if a GRAVFILE file is being saved for future IMPACT runs, the value of GRIDSAVE for the GRAVITY phase must be set to 1.

JPRINT Debug print level for the GRAVITY phase, assuming values of 0, 1, or 2. The default level is zero, which echoes the input stream and prints the final results. A setting of 1 produces a printout of all the applicable run parameters, plus a listing of the integrated gravimetric spectral density at each alongtrack frequency gridpoint. A setting of 2 produces, in addition, a listing of the undulation spectral density and the major terms contributing to it at each grid point in the two-dimensional frequency domain.

As an additional function for this parameter, setting it to a negative value will cause the program to perform initialization only, and then halt; this can be used to check the validity of a control stream without executing it.

4.2.3 Running the IMPACT phase

The impact phase is used to evaluate the statistics of miss associated with a given survey and trajectory. If both the GRAVITY and IMPACT phases are included in a single run, the GRAVITY phase will write temporary files of residual geoid undulation spectral densities. The GRAVFILE parameter should not be specified in the input since the file cannot be saved, for reasons discussed in Section 4.2.2 under the heading Selection of Outputs.

When the IMPACT phase is to be run using a file generated by the GRAVITY phase in an earlier run, the GRAVFILE parameter must be used to indicate the filename. This option allows the user to obtain impact miss statistics for the same survey but for different trajectories. However, this requires that there be one MULTISENS run for each trajectory, using the same GRAVFILE in the multiple runs. In order to specify the residual gravity field, the procedure followed in a single MULTISENS run, including both GRAVITY and IMPACT, can be imitated. This requires the production of five separate GRAVFILE files with five different runs of MULTISENS using MODE=PSD or MODE=COV and with the values of SAMPLEA, SAMPLEC, M1, and M2 as given in Table 4.2-1. The results of the five IMPACT runs must then be combined in the same way as the gravimetric covariance contributions.

Alternatively, a single GRAVFILE file may be used. The choice of values

SAMPLEA = SAMPLEC = 6.75
M1 = M2 = 100

in the GRAVITY run which produces GRAVFILE yields IMPACT results which in most cases differ by less than two percent from the full five-scan computation.

The IMPACT phase also needs as input the inflight transfer function file generated by the TRANSFER phase. The TRANSFER phase may be included in the same run, or the file may have been saved from an earlier run. If both TRANSFER and IMPACT are included in the same run, omission of the TRANSFILE parameter will cause the program to pass a temporary file between the two phases and to delete this file at the end of the run. When TRANSFILE is specified in such a run the file will be retained after completion of the run. This is in contrast to GRAVFILE, which cannot be retained after a run including both GRAVITY and IMPACT.

Only one input parameter applies to the IMPACT phase; this is the real scalar WAVELONG, which indicates the maximum wavelength to be included in the evaluation of miss statistics. Its units are nautical miles. This parameter must be positive, and defaults to 2700.

4.2.4 Sensitivity Runs

The input character scalar parameters SENSITIVITY and VARIABLE and the real vector parameter VALUE may be set by the user to cause the program to perform a sensitivity study. The program will conduct a series of runs for which the variable indicated in the parameter VARIABLE will sequentially assume each of the values listed in the vector VALUE. The program can then tabulate and plot the values of the computed error statistics against those of the independent variable selected. SENSITIVITY may be set to any one of the three values RMS, IMPACT, and AREAMEAN. These values are used to indicate the

dependent variable in the sensitivity study. (Note that a sensitivity run can also be indicated by entering a value for the VARIABLE parameter, as discussed below.)

Setting SENSITIVITY=RMS will provide the sensitivity of the statistics of the point values of the residuals in the gravimetric quantities listed in GRAVERR. It requires or forces the MODE parameter to the value RMS.

SENSITIVITY=IMPACT provides sensitivities of the statistics of both point value postsurvey residuals and impact miss to the independent variable. This option also requires or forces MODE to the value RMS and in addition adds the option IMPACT to the TYPE_OF_RUN specification if it was not included in the control input. Note that an inflight transfer function file is needed for such a run. This file may be precomputed or, alternatively, may be produced in the same run by including the execution of the TRANSFER phase in TYPE_OF_RUN.

SENSITIVITY=AREAMEAN will force the MODE setting to AREAMEAN and will generate a sensitivity study of the statistics of the residuals in spatially averaged gravimetric quantities. Only gravimetric statistics are available in this mode; inclusion of the IMPACT option in the TYPE_OF_RUN specification will generate an error message and terminate the program.

The parameter VARIABLE may assume any one of the following values:

- For all survey options: ALTITUDE, AREASIZE, EL, TRKANGL. (Note that ALTITUDE is valid only in a non-IMPACT run, since all IMPACT runs would produce identical miss statistics. Also, VARIABLE=AREASIZE is valid only in SENSITIVITY=AREAMEAN runs.)

- For runs which include the ALT1 survey: ALAM, ALTVAR, BETAC, CA, HEITALT, ORBINC, SAMPALT, SIGMAC2, TRKSP.

- For runs which include the ALT2 survey: ALAMP, ALTVARP, BETAC, CAP, HEITALTP, ORBINCP, SAMPALTP, SIGMAC2, TRKSPP.

- For runs which include the SST survey: LOHEIGHT, LOORBINC, SSTLONG, SSTNOISE, SSTSP, SSTTRKSP.

- For runs which include the GRAVLAND survey: CG, GRAVXTENT, SE, SN.

- For runs which include the GRAVSHIP survey: BETAC, CG, GRAVXTENT, QQ, SE, SIGMAC2, SN.

- For runs which include any one of the gradiometer surveys: GRADSP, GRADXTENT, HEIGHT, SAMPINT, SPEED.

If the input parameter VARIABLE is set, but SENSITIVITY has not been assigned a value in the control input file, SENSITIVITY=IMPACT is assumed.

A maximum of 20 and a minimum of three values must be entered in the vector VALUE. The number of values entered will determine the number of points to be computed on the sensitivity curve. Their order is immaterial but each of the values has to satisfy the numerical contraints active on the values of the variable indicated by VARIABLE.

In a sensitivity run, any value assigned directly to the independent variable by the control file will be ignored; for example, given the control input:

```
TYPE_OF_RUN = (GRAVITY,IMPACT)   TRANSFILE=TRANS.DAT
SURVEY = (ALT1,GRAVSHIP)
SENSITIVITY=IMPACT          VARIABLE=TRKSP
VALUE=(10.E0,20.E0,40.E0,60.E0,80.E0,90.E0)
TRKSP=50.E0;
```

the program will assign, in successive calculations, values of 10, 20, 40, 60, 80, and 90 nautical miles to the variable TRKSP, and the specification TRKSP=50.E0 will have no effect.

4.2.5 File Format Requirements

There are four files which may be required as input by the phases of the program: a trajectory file for the evaluation of the inflight transfer function in the TRANSFER phase, files describing the NUMERIC gravity model option used in the GRAVITY phase, the geoid undulation error spectral density file produced by the GRAVITY phase and used by the IMPACT phase, and the inflight transfer function file read by the IMPACT phase. The formats of these files are described below.

Trajectory File

The standard trajectory file consists of a series of unformatted records, one corresponding to each time point on the trajectory. Each record consists of ten double precision floating point words, in the sequence: TIME, POS(3), VEL(3), ACC(3). The values of TIME should be in seconds from launch; the three vectors following represent the position, velocity, and specific force at this instant in the flight, and should be expressed in MKS units. The last three entries in each record, ACC(3), are ignored by the program since the error equations (based on a linearization about the nominal trajectory) do not involve specific forces. The velocity vector is used only for the determination of stepsizes in the propagation of the solutions of the differential equations.

The coordinate system for the vectors should be inertial and earth-centered, with the three axes pointed at

the Greenwich meridian, the Greenwich meridian plus 90 degrees, and the North pole, respectively, at the time of launch. The file should contain as many records as needed to define the trajectory from launch to impact, including records corresponding to the launch and impact times. The records should be in increasing order of TIME.

USERFILE Files

Each of the undulation spectral density files used to define the NUMERIC gravity model contains an unformatted header record, followed by a series of unformatted data records.

The header record contains four variables: N1, N2, R1, and R2, which are defined as follows:

N1 is an INTEGER*4 variable which indicates the number of data records to follow. The number of data records will be N1+1, which may be indexed from zero to N1.

N2 is an INTEGER*4 variable which indicates the length of each of the data records to follow. Each data record will contain a vector of 2(N2)+1 REAL*4 elements, which may be indexed from -N2 to N2.

R1 is a REAL*4 variable which indicates the maximum frequency (in radians per meter) covered by the table in the east direction.

R2 is a REAL*4 variable which indicates the maximum frequency (in radians per meter) covered by the table in the north direction.

Each of the data records which follow contains a vector of REAL*4 values of the undulation spectral density for a fixed east frequency, and for equally spaced north frequency values ranging from -R2 to R2. The east frequencies associated with each data record range in equal intervals from zero to R1,

beginning with the zero-frequency record. Thus if the records are indexed from zero to N1 and the data values on each record are indexed from -N2 to N2 as described above, the J-th value on the I-th record will correspond to an east frequency of I×R1/N1 and a north frequency of J×R2/N2. Only the nonnegative east frequency values need be supplied, since the value associated with frequencies W1 and W2 must be the same as that associated with frequencies -W1 and -W2.

Geoid Undulation Error Spectral Density File

The undulation spectral density errors are written by the GRAVITY phase either to a temporary file or to the filename entered in the GRAVFILE parameter of the input stream. (For MODE=RMS or AREAMEAN or for sensitivity runs, the file is written several times, each time destroying the previous version.) The file is unformatted and consists of a header record followed by a series of data records.

The header record contains five variables, in the order: DATE, M1, M2, SAMPLEA, and SAMPLEC, where DATE is a CHARACTER*20 string containing the date and time of creation of the file, M1 and M2 are INTEGER*4 variables corresponding to the input parameters with the same names, and SAMPLEA and SAMPLEC are REAL*8 variables corresponding to the input parameters with the same name.

The header is followed by M1+1 data records, indexed from zero to M1; each data record consists of two INTEGER*4 variables N1 and N2, followed by a vector of 2(N2)+1 REAL*8 values. The integer N1 is simply the record number, and has values from zero to M1; N2 is the record length indicator, and can be used to read the record using the statement:

If the user has selected an altimeter and/or gravimeter survey, two parameters describing the ocean current model will be displayed in the list format. The user can make changes as described in the input format section above.

If the MULTISENS run is to be a sensitivity run, the user will then be prompted for further information necessary for the run. The first question is to define the type of sensitivity. The valid sensitivities are:

- RMS - Sensitivity statistics of point values are computed.
- AREAMEAN - Sensitivity of spatial averages computed. The user will be further prompted for the AREASIZE parameter. The default is 60.
- IMPACT - Sensitivity of impact error statistics are computed.

Next the user should enter the parameter which is the independent variable in the sensitivity run. The variables available as the independent variable are a subset of all input parameters. The valid sensitivity variables are:

ALAM	CG	LOORBINC	SN
ALAMP	EL	ORBINC	SPEED
ALTITUDE	GRADSP	ORBINCP	SSTLONG
ALTVAR	GRADXTENT	QQ	SSTNOISE
ALTVARP	GRAVXTENT	SAMPALT	SSTSP
AREASIZE	HEIGHT	SAMPALTP	SSTTRKSP
BETAC	HEITALT	SAMPINT	TRKANGL
CA	HEITALTP	SE	TRKSP
CAP	LOHEIGHT	SIGMAC2	TRKSPP

For a specific run, the acceptable variables may be a subset of those listed above, depending on what surveys are included in the run. If the variable entered is not used in the current

- NEGRAD - north-east gravity gradient
- NUGRAD - north-up gravity gradient
- EUGRAD - east-up gravity gradient

Up to 10 quantities can be entered, separated by valid delimiters, with UND being the default value. The order of the inputs will be retained.

A list of general MULTISENS input variables is then presented in the table form of Figure 4.3-2. After this, the type of survey is to be entered with valid surveys being:

- SST - Satellite to satellite doppler
- ALT1 - First satellite altimeter
- ALT2 - Second satellite altimeter
- GRAVSHIP - Ocean-based gravimeter
- GRAVLAND - Land-based gravimeter
- DRAPER - Draper gradiometer
- DRAPERC - Carouselled Draper gradiometer
- BELL - Bell gradiometer
- BELLC - Carouselled Bell gradiometer
- GOAL - Goal gradiometer
- GRAD - User specified gradiometer

The full survey name must be entered. Abbreviations are not acceptable for survey input. Entering a blank line will give the default value of <u>no surveys</u>. Up to five different surveys may be entered with two restrictions. Only one type of <u>gravimeter</u> survey is allowed; only one type of <u>gradiometer</u> survey is allowed.

Only one model should be entered with BASELINE being the default value. If the NUMERIC model is selected, the user will be prompted for files cointaining the model. The contents of these files are specified in sections 4.2 and 4.2.5. Up to three filenames can be entered. If the filenames cannot be entered on a single line, the continuation character > can be used between filenames to continue the input on successive lines. If the USER model is chosen, there are some additional considerations. The USER model requires the inclusion of a routine USRFUNC describing the model. If this routine is not included in the system version of MULTISENS, MULTIJOB will create a control input file which will access the system version of MULTISENS and not the one containing the new USRFUNC. MULTIJOB can still be used to created a job stream; however, it should not be submitted automatically but rather saved. The saved file can then be edited and the RUN MULTISENS line should be changed to run the executable file which contains USRFUNC. A user unfamiliar with the editor should consult Volume 3A of the VAX/VMS Reference Manuals.

The user is then prompted to enter the gravimetric quantities whose statistics are to be computed. Valid quantities are:

- UND - undulation of the geoid
- NORTHVD - north deflection of the vertical
- EASTVD - east deflection of the vertical
- ANOMALY - gravity anomaly
- NNGRAD - north-north gravity gradient
- EEGRAD - east-east gravity gradient
- UUGRAD - up-up gravity gradient

The default value for MULTIPLOT is a GRAVITY run. Any combination of the three may be entered. From this point on, the program prompts for information dealing with those phases the user has selected in the following order: inflight transfer function, gravity residual analysis, and impact error computation. The phase for which the questions are being asked always appears at the top of the screen.

TRANSFER

If TRANSFER was selected, the next set of questions deals with the inflight transfer function computation. The questioning begins with a table of input parameters which the user can change as described in the input format section above. Next the program asks for a trajectory file name. This file must already exist. There is no default trajectory file; the user must supply one. Finally the user is asked whether the transfer function calculations are to be saved permanently. If so, a filename must be entered since there is no default filename. This file need not already exist as it will be created by MULTISENS.

GRAVITY

The GRAVITY residual questions will next appear if the user has selected GRAVITY. This begins with the selection of a gravity model. The valid models are:

- BASELINE
- ACTIVE
- AWN
- NUMERIC
- USER.

After the command is entered, the screen will clear and display an initial message giving some general information. At this point the user is asked whether the input verification mode is desired. This mode echoes back what the user has entered. The user is asked whether this information is correct before the program will proceed. An example of the verification mode output is shown in Figure 4.3-3. If the user is not satisfied with the entry, the prompt will reappear for the user to enter another value. If the user desires this mode, a YES should be entered. If the faster nonverification mode is desired, the user can enter a NO or RETURN. The validity check on the input parameters is always done and does not depend on whether the verification mode is on.

```
What type of run do you want? (TRANSFER, GRAVITY, or IMPACT)
                                                     -------

Enter one, two, or all three: TRAN,GRAV,IMP

The types of run you have selected are:
TRANSFER
GRAVITY
IMPACT

Is this correct? (YES or NO)
```

Figure 4.3-3 Sample of Input Verification Mode Responses.

The questions presented from this point on in the program will gather information to be used in constructing the job stream. The first will ask for the type of MULTISENS run the user wants. Valid inputs at this point are:

- TRANSFER - The inflight transfer function is computed.
- GRAVITY - The gravity residuals are computed.
- IMPACT - Downrange and crossrange errors and CEP are computed.

QUIT - This command will terminate the program with no control input file created.

BACK - The program will reset back to a previous prompt so that previously entered inputs may be modified. Successive BACK commands will eventually bring the user back to the begining of the program. The BACK command resets most of the variables except for those displayed in the table form of Fig. 4.3-2, which retain their previous values.

HELP - This command will print out a brief message to explain in more detail what needs to be entered. It is more useful in obtaining a description of the appropriate syntax than in obtaining a description of the options available. For a description of the options, the MULTISENS user's manual should be consulted (section 4.2).

If a default answer is applicable at any point, the default value is shown underlined in the list of acceptable values. Not all inputs have a default value assigned to them. In some cases the default value implies none; for example, <u>no survey</u> in the <u>type of survey</u> question. If there is no default value a message is printed to that effect. For the questions that are to be answered with a YES or a NO, the default is always NO. The only acceptable entries for YES or NO questions are YES, NO, BACK, QUIT, HELP, <CR>, or beginning substrings of these words. Any other inputs will cause the program to reprompt the user.

4.3.3 Program Execution

The program is executed by entering the command:

RUN MULTIJOB

This section lets you change the current values of the input parameters which affect the SST survey. The input variable will appear with its current value displayed. If no changes are desired enter a carriage return <CR>. If a new value is desired, enter the new value and a <CR>.

VARIABLE	UNITS	CURRENT VALUE	NEW VALUE
LOHEIGHT	KM	150.0	
LOORBINC	Deg	86.0	
SSTFUNC		0	
SSTLONG	Deg	0.0	
SSTNOISE	MicroM/SEC	100.0	110.0
SSTSP	SEC	10.0	
SSTTRKSP	NM	30.0	

Do you wish to make any changes? (YES or NO)

Figure 4.3-2 Sample of Table Input Parameter Prompting

with the highest version number is used. Semicolons should not be used to separate version numbers from the rest of the filename since the semicolon is used as an input delimiter in this program. Its use would cause the program to ignore the version number. Volume 2A of the VAX/VMS Reference Manuals contains a complete description of valid file specifications. If the file is required as an input to MULTISENS, it is checked for existence. If the file is an output of MULTISENS, the name is checked only for a valid file specification.

In most places, abbreviations of three or more letters will be accepted. Exceptions to this are filenames and survey specifications.

When running the program, the user can enter the following special commands at the beginning of any input line.

The second input format consists of entering values to a table of input parameters. As the display develops on the screen, the variable, its units, and its current value will appear one line at a time. If the current value is satisfactory a blank line should be entered by pressing the RETURN key <CR>. The next variable will then appear for inspection. If a different value is desired, the value should be entered at the current cursor position in the NEW VALUE column with no more than 20 characters; the length of the underlining for the column is 20 characters. The delimiter characters mentioned above are not active for table input. After all the variables for the section have appeared, the user will be asked if any further changes are desired. If a YES is entered, the display will reappear with any new values replacing the old in the current value column. The display will again develop line by line, with the user able to make further changes to the variables. The program will continue when the user enters a NO to the further changes prompt. If after a number of changes the user wishes to see the original values, HELP should be entered in the new value column. This will display the original values for the parameters in the current table. Figure 4.3-2 shows what a completed display looks like with some changes entered.

When a filename is entered, the name must include all qualifiers to identify the file uniquely. A complete file specification has the following format :

device:[directory]filename.type.version

The punctuation marks (colons, brackets, periods) are required to separate the various components of the file specification. If the input file is on the same directory as the user, only the filename and type need to be specified. The default filetype is DAT, and if no version number is given, the file

a single delimiter. The '!' character is used to end the current input line. Any characters entered on the same line after a '!' will be ignored and therefore can be used as a comment. The '>' delimiter is similar to the '!' in that the current input line is ended but has the added feature of allowing the input to continue on the next line. Any characters entered after the '>' on the same line will be ignored. If the prompt requires only one answer and more are entered, separated by valid delimiters, only the first value will be used. In multiple-entry questions, second and subsequent appearances of the same value will be ignored. Examples of various input line formats are illustrated in Fig 4.3-1. As shown, the group of lowercase x's is ignored and does not constitute part of the input.

```
The types of surveys available are: (ALT2,      ALT1,     GRAVSHIP, GRAVLAND,
                                     SST,       DRAPER,   DRAPERC,  BELL,
                                     BELLC,     GOAL,     and GRAD)

Note:  The default value is no survey.

     Enter full name of survey, no abbreviations accepted

     Any combinations of surveys are acceptable with the following exceptions

         1.  GRAVLAND and GRAVSHIP are mutually exclusive

         2.  Only one type of gradiometer allowed (DRAPER,
             DRAPERC, BELL, BELLC, GOAL, or GRAD)

Enter as many as desired: SST,ALT1; ALT2 (GRAVLAND>
 DRAPER ! xxxxxxxxxxxx

What gravity quantities do you want included?
         (UNDULATION, NORTHVD, EASTVD, ANOMALY, NNGRAD, EEGRAD,
          ----------           UUGRAD, NEGRAD,  NUGRAD, and EUGRAD)

Enter up to 10 quantities: NORTHVD, EASTVD > xxxxxxxxx
ANOMALY(UUGRAD)EEGRAD>
EASTVD, EUGRAD ! THE SECOND EASTVD WILL BE IGNORED AS WELL AS THIS COMMENT
```

Figure 4.3-1 Samples of Individual Question Inputs

4.3.1 Requirements and Limitations

This section of the user's manual is concerned only with the execution of the MULTIJOB program. Since it is the object of this program to create the control input file for the MULTISENS program, the user should be familar with the inputs required by MULTISENS. The MULTISENS user's manual is given in section 4.2.

No files are used as input to MULTIJOB; however, since some configurations of MULTISENS require input files, these files should exist prior to the execution of MULTIJOB. MULTIJOB will verify that the files exist, but will not check to see whether the information they contain is in the correct format. The requirements for these files are given in section 4.2

4.3.2 Input Formats

There are two general ways in which the user is prompted for information. The first method involves questions which appear one at a time on the screen, to which the user responds before the next question appears. A sample of this is shown in Fig. 4.3-1.

Delimiter characters are available to separate the responses to questions which call for more than one entry. These delimiters are ' '(blank), ',', ';', '/', '(', ')', '!', or '>'. The '!' and '>' are special delimiters which are discussed in more detail below. Since the semicolon is used as a delimiter it cannot be used as part of a filename specification. If a version number is required in a filename it should be separated from the rest of the filename by a period. Two or more delimiters appearing together (e.g. ',;'), are treated as

The function USRFUNC will be invoked by function FFUNC for runs in which the MODEL=USER option is selected. It should be defined as:

REAL*8 FUNCTION USRFUNC (W1,W2)

where the arguments W1 and W2 are REAL*8 scalars containing the east and north frequency values respectively, in units of radians/meter. The value returned should be the modeled undulation spectral density at those frequency values, in units of $m^2/(cycles/m)^2$.

Either of the standard functions MARKOV2 and AWN may be used as a guide in constructing a USRFUNC module.

4.3 RUNNING THE MULTIJOB PROGRAM

The MULTIJOB program is an interactive tool to create a job stream for the MULTISENS program. It leads the user through the various options available in MULTISENS using previous inputs to update available options. If a previous selection has made another option unavailable, that other option will not be presented. The program also checks the inputs for consistency and validity. A control input file is created which can automatically be submitted and/or saved under a user specified filename. If the file is saved, it can be submitted as a batch job by the SUBMIT command or the program can be run at a terminal using the execute procedure (@) command. When @ is used as a prefix to a filename (i.e., @FILENAME) the lines of the file are taken to be commands which are to be executed as if they were entered at a terminal. For more information on the execute procedure command the user should consult Volume 2A of the VAX/VMS Reference Manuals. The saved file can then be edited using the EDT editor to make further changes, without rerunning MULTIJOB.

FLAG	is used as both an input and output argument. Its type is INTEGER*4. On entry to the subroutine, FLAG indicates the function to be performed on this call; for FLAG=0 the subroutine should open the file, for FLAG=1 the subroutine should close the file, and for FLAG=2 the subroutine should read the next trajectory record and return the contents in the last three arguments. Before returning from the subroutine, the argument FLAG should be set to one of two values: 0 (zero) if the I/O function requested was performed correctly, and 10 if an end-of-file was encountered on a read operation (i.e., when the input value of FLAG was 2).
TIME	is a REAL*8 scalar output argument which is used only for a read operation; i.e., when the input value of FLAG is 2. It should be filled with the time value read from TRAJFILE. Units are seconds from launch.
POS	is a REAL*8 three-vector output argument, which is used only for a read operation. It should be filled with the position vector read from TRAJFILE, converted if necessary to an earth-centered inertial Cartesian system with the three axes oriented to Greenwich, Greenwich + 90 deg, and the North pole at time of launch. Units are meters.
VEL	is a REAL*8 three-vector output argument, which is used only for a read operation. It should be filled with the velocity vector read from TRAJFILE, converted if necessary to the same coordinates as POS. Units are meters/second.

The standard version of GETTRAJ may be used as a guide in constructing a substitute.

of J and K, respectively, loop over the east and north frequencies, and the index L is associated with downrange miss (L=1) and crossrange miss (L=2).

4.2.6 Interfaces for User-Written Subroutines

There are two subroutines in the program which are designed so that the user may replace them to suit a particular application. The first of these is GETTRAJ, whose replacement will enable the TRANSFER phase to use an input trajectory file in other than the standard format (which is described in section 4.2.5). The second is USRFUNC, which will be called in the GRAVITY phase to provide undulation spectral density values as a function of east and north frequency if MODEL=USER is specified in the input; this enables the user to run with a gravity model other than those supplied with the program.

GETTRAJ is a subroutine used to open, read, and close the input trajectory file. Its calling sequence is:

```
CALL GETTRAJ (ITRAJ,TRAJFILE, FLAG,TIME,POS,VEL)
```

and the arguments are defined as follows:

ITRAJ — is the FORTRAN unit number assigned to the trajectory file. The subroutine should not change the value of ITRAJ, but should use it in the OPEN, READ, and CLOSE statements of the program. It should be declared INTEGER*4.

TRAJFILE — is the filename of the input trajectory file. The calling program will set this value according to the TRAJFILE specification in the input stream. The subroutine should not change this parameter, but should use it in the OPEN statement for the trajectory file. It should be declared CHARACTER*(*).

```
READ (IGRAV) N1,N2, (REC(J),J=-N2,N2)
```

Normally N2 is equal to M2; they may differ, however, if the parameter GRIDSAVE is set to a value less than 1. (Such a file is not suitable for running the IMPACT phase.) In general, the value indexed J in the record indexed I contains the geoid undulation spectral density value for frequencies WA alongtrack and WC crosstrack, where:

$$WA = \pi I/[(MPERNM)(SAMPLEA)(M1)]$$

$$WC = \pi J[(MPERNM)(SAMPLEC)(N2)]$$

where N2 is the record length indicator of the Ith record and MPERNM is the number of meters in a nautical mile (1852). The units of WA and WC are in radians per meter, and those of the spectral density values are in $m^2/(cycles/m)^2$.

TRANSFILE File

The computed inflight transfer function is written to an output file whose name is specified as the TRANSFILE parameter in the control input. If the TRANSFILE parameter is missing from the control input, a temporary file is created, which is deleted at the end of the run. The file consists of a single unformatted record, of the form:

```
NF, F0, ((((G(I,J,K,L),I=1,2),J=1,NF),K=1,NF),L=1,2)
```

where NF is an integer matrix dimension (equal to NFM1+1), F0 is the double precision reciprocal of the input parameter LAMBDA0 (converted to inverse meters), and G is a single precision complex inflight transfer function table, which is dimensioned (2,NF,NF,2). The index I indicates whether north frequencies are positive (I=1) or negative (I=2). The values

setup of the MULTISENS run, a message will inform the user that the variable is not valid for this run and a new one will have to be entered. Once an acceptable variable has been entered, the values for that variable must be entered. They should be separated by any of the delimiters and > used if the input must be continued on the next line. Up to 20 values may be entered but a minimum of three values is required.

If plotted output from the sensitivity run is desired, the user has a choice of having hard copies made during the batch job, or of storing the information in a file to obtain plots later using MULTIPLOT. If the plot information is to be saved, a filename can be entered or the default name TMPPLTMLT.DAT is used. The filename is checked only to verify that it has a valid file specification.

If this is not to be a sensitivity run, the user will be prompted for the type of output for the run. The acceptable modes are:

- PSD - Power spectral density
- COV - Covariance
- RMS - Point values
- AREAMEAN - Area mean values.

Only one value should be entered with RMS being the default value. If AREAMEAN is selected, the user will be further prompted for the area size. The default value is a series of runs that yield the statistics of the 5, 15, 60, and 300 min means. If these are not acceptable, only one value can be entered for the area size prompt.

If either PSD or COV is entered, the user will be allowed to change the integration region parameters (described in section 4.3.2) in the table input format. Also, if plotted output is desired, the user has a choice of having hard copies made during the MULTISENS program or of storing the information in a file. If the information is to be stored, a filename should be entered or the default name TMPPLTMLT.DAT is used. If hard copies are selected, the user will enter the types of plots. Some plots are not available if the mode PSD has been selected. The valid plots are:

- PSD - Power spectral density plots available with a MODE of PSD or COV
- COH - Coherence plots available with a MODE of PSD or COV
- COV - Covariance plots available only with a MODE of COV
- COR - Correlation plots available only with a MODE of COV
- 3-D - Surface plots available with a MODE of PSD or COV
- CONTOUR - Contour plots available with a MODE of PSD or COV.

Depending on the parameter MODE, up to 6 values can be entered. The defaults are COV when MODE=COV and PSD when MODE=PSD.

Following the selection of plotting options, the last GRAVITY phase inputs are entered. These consist of the parameters describing the geometry and sensor error models for the various surveys previously selected by the user. The parameters are presented in the table input format. Any changes can be made as described in the section on input formats (section 4.3.2). Also, if GRAD has been selected, the user will be

required to enter the values for both the red and white noise parameters.

IMPACT

The IMPACT error computation questions are next. These questions are asked only if IMPACT was included in TYPE_OF_RUN or a SENSITIVITY of IMPACT was selected. Depending on the current program setup, the user may be asked for a file containing the inflight transfer function table (TRANSFILE) and/or a file containing the geoid undulation error spectrum (GRAVFILE). If requested, these files must already exist. The parameters for the IMPACT phase will then be presented in the table input format.

JOB SUBMITTAL

The final section of the program deals with the actual creation of the job stream. The user is asked whether the job is to be submitted and whether the control input file is to be saved. The control input file created does not have to be submitted in order to be saved. If the file is saved, it can subsequently be edited to incorporate changes in the job stream. If the file is to be saved, a filename should be entered or the default TMPSMLT.COM is used. If no filetype is specified, the default COM is assumed. The filename is checked only for a valid file specification. If the job is to be submitted, a message stating that it has been submitted will be printed. The user is then asked whether the MULTIJOB program is to be restarted in order to create another control input file. If not, the program will terminate. If the program is restarted, the input variables displayed in the table input format are not reset to their original values but contain the values that were used in the previous run. The original values can be inspected by entering HELP in the NEW VALUE column.

If the file created by this program is saved, it can be submitted as a batch job via the SUBMIT command. The file can also be run at the terminal with the execute procedure (@) command. For more help with the @ command the user should consult Volume 2A of the VAX/VMS Reference Manuals.

4.3.4 Error Messages

The MULTIJOB program contains several error traps to detect and flag some common errors. The two major types of errors detected are:

- Input errors - These occur when an undecipherable word or number has been entered. These are flagged as invalid options or nonnumeric values and the user is prompted to reenter the values.
- Validity errors - These occur when a value for an input parameter is out of the valid range for that variable. A message will be printed and the user prompted to enter a new value.

Invalid file names and nonexistent files are included in the input errors. Each of these is flagged with the messages:

```
*** Error in locating file.  Check spelling or
    include path name if file is in another directory.
```

```
*** File specification error.  Check for illegal
    character.
```

The first may occur if the filename is misspelled, or if the file is in another directory and the path name was not included. The second message occurs when an illegal character is contained in the file name or when the file specification is not in the

correct format. The user should consult Volume 2A of the VAX/VMS Reference Manuals for further information on file specification.

Another type of input error is an invalid or undecipherable entry. These are flagged with a message of the form:

```
XXYYZZ is invalid, reenter
```

If this should occur, only that one invalid value need be re_ entered. Similarly, if the input is not a valid number, the message

```
*** Variable  TRKANGL has a nonnumeric value  10.R2
   Reenter value:
```

will be printed. When this is encountered, the user should reenter the corrected value.

Among the validity errors are those detected by MULTISENS. These include errors of conflict of parameters and values out of range. The first includes such errors as trying to get a sensitivity of AREAMEAN when IMPACT has already been selected. This would produce the message:

```
Cannot have IMPACT with a SENSITIVITY of AREAMEAN
```

The second type, value out of range, would produce a message similar to:

```
*** Variable ALTITUDE has an invalid value of  -200
 Valid values are: ALTITUDE => 0.0
 Reenter value:
```

The user need only reenter an acceptable value for the variable. No default values exist at this time.

4.4 RUNNING THE MULTIPLOT PROGRAM

MULTIPLOT is an interactive program used to display graphical output on the Lexidata color graphics terminal, based on information stored during a MULTISENS run. MULTIPLOT allows the user to change scales on the plots and select which ones to present. It cannot change MULTISENS parameters or recompute the plots.

4.4.1 Preprocessing Requirements and Limitations

MULTIPLOT requires as input a file containing the plot information. This file must have already been created by the MULTISENS program. Information stored in this file tells what options were used in the MULTISENS run and what plots will be available during the execution of MULTIPLOT.

Since MULTIPLOT also requires the use of the Lexidata color graphics terminal, the user should check that the graphics terminal is available prior to running the program. Also, the toggle switches located on the trackball unit should be in the OFF position. The program will execute even if the graphics terminal is not available, but no plots will be generated on the screen. The plot menu will appear and its functions, except for the zoom, will be valid. The user can then generate hard copies if so desired.

4.4.2 Program Execution

MULTIPLOT is executed by entering the command

RUN MULTIPLOT

The user will first be prompted for a file name containing the plot information previously created by the MULTISENS program. The default filename is TMPPLTMLT.DAT.

After the file name has been entered, the screen will clear and information concerning the MULTISENS run that created the file will appear at the top of the VT100 screen. This information will remain at the top of the screen while the current plot file is used. It will also appear on the graphics terminal under the plots, except when contour or surface plots are being displayed. The prompt

Enter command:

will then appear. Whenever this prompt is on the screen, the user has the choice of which plots are to be displayed. Entering HELP will present a list of acceptable commands and a brief description of their use. The available commands are different for the two general categories of plots: sensitivity or gravimetric statistics. The sensitivity plots display impact errors, point values, or spatial average sensitivities to the independent variable. The statistics plots can display covariances, correlations, power spectral densities, coherences, or two-dimensional power spectral densities as surface or contour plots, depending on the parameters used in the MULTISENS run.

There are three main categories of commands that the user can enter. These are

- General commands available during both sensitivity and statistical plotting

- Sensitivity plot commands
- Statistical plot commands.

A description of each of the commands available in each of these categories follows. If an invalid command is entered the program will reprompt the user for another command. The general commands available are:

QUIT - Program execution is terminated.

BACK - This will reset the program back to a previous location so that different options can be selected. A BACK entered at the "Enter command: " prompt will allow the user to enter a new file by returning to the enter file name prompt.

HELP - This command will print out a brief message to explain in more detail some aspects of the current question. This command is not present at all locations.

Commands which are available only for sensitivity plotting are:

IMPACT - The downrange, crossrange, and CEP sensitivity curves are displayed. This is valid only if MULTISENS was run with SENSITIVITY = IMPACT.

RMS - The point value sensitivity curves are plotted. This is valid only if MULTISENS was run with SENSITIVITY = RMS or SENSITIVITY = IMPACT.

AREAMEAN - The sensitivity of spatial average curves is displayed. It is valid only if MULTISENS was run with SENSITIVITY = AREAMEAN.

The commands available only for the gravimetric statistical plots are:

COVARIANCE - The covariances of the gravimetric quantities specified in the MULTISENS run are displayed. This requires that MODE = COV in the MULTISENS run.

CORRELATION - The correlations of the gravimetric quantities specified in the MULTISENS run are plotted. This requires that MODE = COV in the MULTISENS run.

PSD - The power spectral densities of the gravimetric quantities specified in the MULTISENS run are displayed.

COHERENCE - The coherences of the gravimetric quantities specified in the MULTISENS run are plotted.

SURFACE/3-D - Either command will create a surface plot of the two-dimensional power spectral density generated by MULTISENS.

CONTOUR - This command produces a contour plot of the two-dimensional power spectral density generated by MULTISENS.

All the plotting commands, except for CONTOUR and SURFACE/3-D, will display the following menu:

<CR>/CLEAR	-	to clear screen and continue
PRINT	-	to produce hard copy
ZOOM	-	to zoom and scroll
SCALE	-	to change axis scaling
BACK	-	to return to command input
QUIT	-	to terminate the program.

The commands in more detail are:

<CR>/CLEAR- This will clear the Lexidata screen. If there is another plot in the series it will be presented, otherwise control will go back to the command input line with the "Enter command: " prompt.

PRINT - A hard copy of the current plot shown on the Lexidata will be printed on the line printer after the program stops.

ZOOM - The user has the ability to expand any area of the plot. After entering ZOOM, the user sets any WHITE switch on the trackball unit to the ON position. The trackball can now be used to position any particular part of the plot on the screen. If a higher magnification is subsequently set, the area expanded will be the upper right corner of the screen. The WHITE switches set the magnification level, with switch E (the first WHITE switch to the left of the BLUE one) having a unity magnification and magnification increasing by two with switch A (rightmost WHITE switch) having a magnification factor of 16. When the user is finished with the zoom facility, all switches should be set to their OFF position and the BLUE switch should be toggled to redisplay the plot menu on the VT100.

SCALE - SCALE is entered if the user wishes to change the type of axis (linear or log) and/or the end points of the scale. After SCALE is entered, the user is prompted to specify the type of axis by entering one or two of the following:

XLOG - The x-axis will have a logarithmic scale

XLINEAR - The x-axis will have a linear scale

YLOG - The y-axis will have a logarithmic scale

YLINEAR - The y-axis will have a linear scale

Any scale which is not affected by an entered command remains the same. A blank line can be entered if the user does not wish to change the type of axis. The user will next be prompted for the minimum and maximum for each axis. The default

value displayed is used if a blank line is entered. If the minimum and maximum for an axis are equal, automatic scaling will be done by the program. The system plotting package will not always use the exact minimum and maximum values as entered as the end points, but the discrepancy will not be significant. If the user changes an axis type to LOG and the current minimum is less than or equal to zero, the minimum is changed to 10^{-8}.

BACK - This command will return the user to the "Enter command:" prompt.

QUIT - This command will terminate the program execution.

For the contour plot, the menu does not contain the SCALE option. For the surface plot, the SCALE option is also not available but in this case it is replaced by an ANGLE option. This lets the user observe the current plot from any viewing angle. The user will be prompted for two angles measured in degrees. The first is an angle in the x-y plane measured counterclockwise from the positive x-axis to the line of sight. The default value for the first plot is -60. The second angle is from the x-y plane to the line of sight. Positive values are above the plane; negative values view from below the plane. The default value for the first plot is 20.

If the user wishes to stop program execution, at any point in the program, QUIT should be entered. If the user wishes to get back to a previous point, the BACK command should be used.

4.4.3 Error Messages

The program has several error traps designed to detect and flag some common errors. If the error is an invalid or undecipherable command, the messages that appear will be of the form:

```
*** Error in input, retry
Invalid command, retry
```

If one of these is encountered, the user should check the spelling or make sure the command is valid for this run of the program. The HELP command could be of assistance.

There are two possible errors concerning the input file specified. These will print one of the following messages:

```
*** Error in opening file FILENAME
    Check for misspelling or incorrect path name

*** Error in reading plot file
    File possibly not created by MULTISENS
```

The first will appear if the file specified (FILENAME) cannot be found, an invalid file specification is entered, or invalid characters are included in the filename. The user should check the spelling and/or specify the path name if the file is not in the current directory. The second error message occurs when the program cannot read the file. This could happen if the file was not created by MULTISENS. The user should make sure that the correct filename was entered. When any of the above messages appears on the screen, the user will be re-prompted to enter a filename. If the user wishes to terminate the program, QUIT should be entered as the filename.

If, during the sensitivity plotting, the user enters a command for which there are no data because MULTISENS did not contain the appropriate option, one of the following messages will appear:

This MULTISENS run did not compute impact sensitivity

This MULTISENS run did not compute sensitivity of
point values

This MULTISENS run did not compute sensitivity of
area means

A warning message which MULTIPLOT may print is

```
*** Scale spans 0.0
    Log scale changed to linear
```

This message is generated for a log scale if the axis would contain zero or negative numbers. The scale affected is automatically converted to a linear scale. If the user still wishes a log scale, then the SCALE option on the plot menu should be used and the upper and lower bounds of the log scale changed to include only positive values.

TABLE 5.2-2

INPUT PARAMETERS FOR GEOEST3

IE	TYPE	DIMENSION	DEFAULT VALUE	RANGE	DESCRIPTION
	CHAR*80				Case Identifier
	INTEGER	2	16	4,8,16,32	Row and Column Dimensions
	INTEGER	2	5	0-N/2	Matrix Row and Column Bandwidths
ı	INTEGER	2	MB**	0-MB	Window Row and Column Bandwidths
'FLG	INTEGER	1	0	(-1)-3	Print Level Flag
'LAG	INTEGER	1	0	-	Not Used In Present Version
)W	INTEGER	2	N/2+1	1-(N/2+1)	Maximum Row and Column Frequency Index
.TA	• REAL*8	2	10^{-6}	0.0-2.0	Row and Column Condition Parameters
'ORE	INTEGER	1	1	0,1	Output Storage File Flag
;	REAL*4	1	10^{-6}	0.0-0.001	Inverse Condition Tolerance
'EPS	INTEGER	1	50	1-100	Number of Inverse Iteration Steps
'S	REAL*4	1	0.001	0.0-1.0	Inverse Iteration Tolerance

ıese parameters should normally be set to their defaul values.
ıA defaults to 0 if MB=N/2.

The inverse iteration is terminated when either the vergence tolerance (CEPS) is reached or the maximum number steps (MSTEPS) is exceeded. The covariance criterion is a sure of the relative rms residual in the frequency domain. ller values of CEPS will require more steps for covergence.

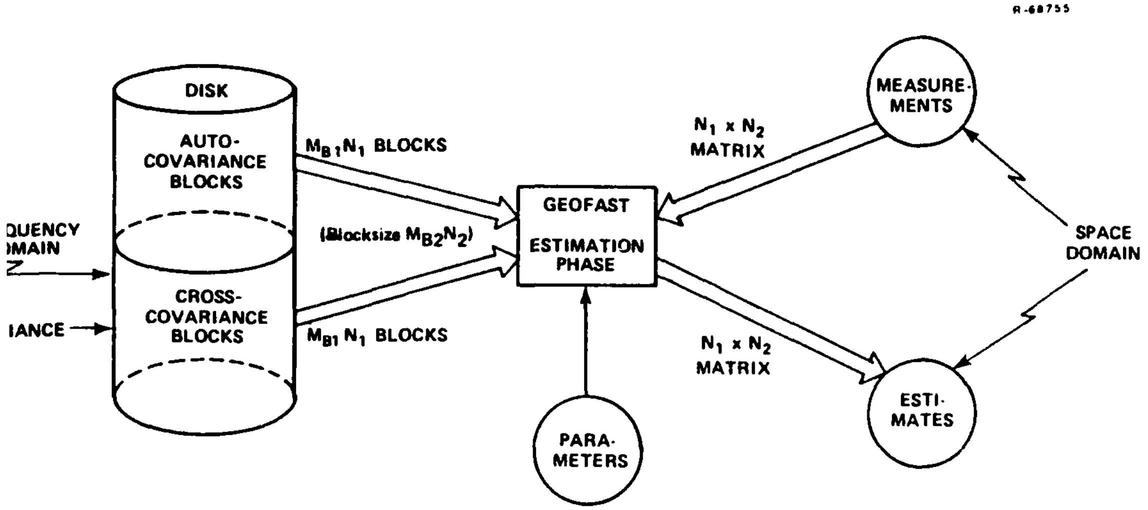


Figure 5.2-4 GEOFAST Estimation Phase

e an alphabetic idenifier and the matrix dimensions. The esent version of the code accepts row and column dimensions 4, 8, 16, or 32 only. This may be increased to any power of o by recompiling the main program (see program documentation). ditional information about certain prameters is given below. ee also Ref. 8).

The condition tolerance parameter (EPS) terminates e Cholesky matrix inversion when any pivot element is less an EPS in absolute value. Increasing the condition paramers (DELTA) will improve the matrix condition but may slow vergence.

The frequency limit parameter (LROW) truncates the equency domain representation at the specified row or column the transformed data matrix.

here DIRECTORY is the name of the current file directory and ILE-NAME is the name of the datastream file containing the arameter values and matrix filenames. All files should reside n the current directory or be explicitly named.

rrors and Diagnostic Messages

Parameter input is checked for errors in the GEOFAST ommand program, but not during execution of GEOCOV3. Incorrect arameter specification may lead to run-time errors. There re no diagnostic messages since there are no error conditions hen the dimensions and bandwidths are consistent with the atrix input. Increasingly detailed printout of intermediate alculations may be obtained by setting IDEBUG = 1, 2, or 3. etting IDEBUG = -1 will suppress printing of the input covari- nce matrix. The number of records written to the covariance utput file is also printed.

5.2.3 The GEOEST3 Program

'urpose

This program is the estimation phase of the GEOFAST stimation algorithm. It computes estimates from the input lata matrix by the minimum-variance method. The calculation s performed in the frequency domain using the covariance tored on disk by the covariance phase (GEOCOV3). The program ay be run with different data sets without changing the co- 'ariances when appropriate. (See Fig. 5.2-4).

'arameter Input

The list of parameters required by GEOEST3 is shown n Table 5.2-2. For each parameter the data type, dimension, efault value, and restrictions are given. Mandatory inputs

where

```
KK = K-1               K=1,..., N1
     K1-2(N1)          K=N1+2,..., 2(N1)

LL = L-1               L=1,..., N2
     L-1-2(N2)         L=N2+2,..., 2(N2)
```

The value of CXZ(K,L) when K=N1+1 or L=N2+1 is arbitrary and may be set to zero.

The elements of the matrix CXZ must be stored as double precision (REAL*8) numbers in column order in an unformatted file. If the matrix CXZ is created by a FORTRAN program as a double precision array and written to disk by a statement of the form "WRITE (IUNIT) CXZ" then the resulting file is in the correct form to be read by GEOCOV3. (See description of subroutine CINPUT in the program documentation for further details). The input covariance matrix is also normally printed.

Program Execution

The GEOCOV3 program may be executed interactively by the RUN command. In this mode the user is prompted for a data stream file containing the parameter input values. Included in this file is the name of the matrix input file and the name of the matrix output file. The required formats can be obtained by creating the data stream file through the GEOFAST command program.

The GEOCOV3 program may also be executed in the batch mode. In this mode the user must create and submit a command file of the form

```
$SET DEFAULT [DIRECTORY]
$RUN GEOCOV3
FILE-NAME
```

TABLE 5.2-1
INPUT PARAMETERS FOR GEOCOV3

NAME	TYPE	DIMENSION	DEFAULT VALUE	RANGE	DESCRIPTION
ID	CHAR*80				Case Identifier
N	INTEGER	2	16	4,8,16,32	Row and Column Dimensions
MB	INTEGER	2	5	0-N/2	Matrix Row and Column Bandwidths
*MBA	INTEGER	2	MB**	0-MB	Window Row and Column Bandwidths
*IDEBUG	INTEGER	1	0	(-1)-3	Print Level Flag
ISWTCH	INTEGER	1	2	1-3	Matrix Symmetry Indicator 1=No Symmetry 2=Symmetric 3=Antisymmetric
*ISTORE	INTEGER	1	1	0,1	Output Storage File Flag
*DELTA	REAL*8	1	0	-	Matrix Condition Parameter

*These parameters should normally be set to their defaul values.
**MBA defaults to 0 if MB=N/2.

Let (I,J) be row and column subscripts for the grid locations of dimension N1 and N2, respectively. Then the cross covariance between X(I1,J1) and Z(I2,J2) is given by a function FXZ(I2-I1,J2-J1) whose arguments vary in the ranges -(N1-1) to +(N1-1) and -(N2-1) to +(N2-1). The input matrix CXZ(K,L) is defined for K=1,...,2(N1) and L=1,...,2(N2) by the formulas:

$$CXZ(K,L) = FXZ(KK,LL)$$

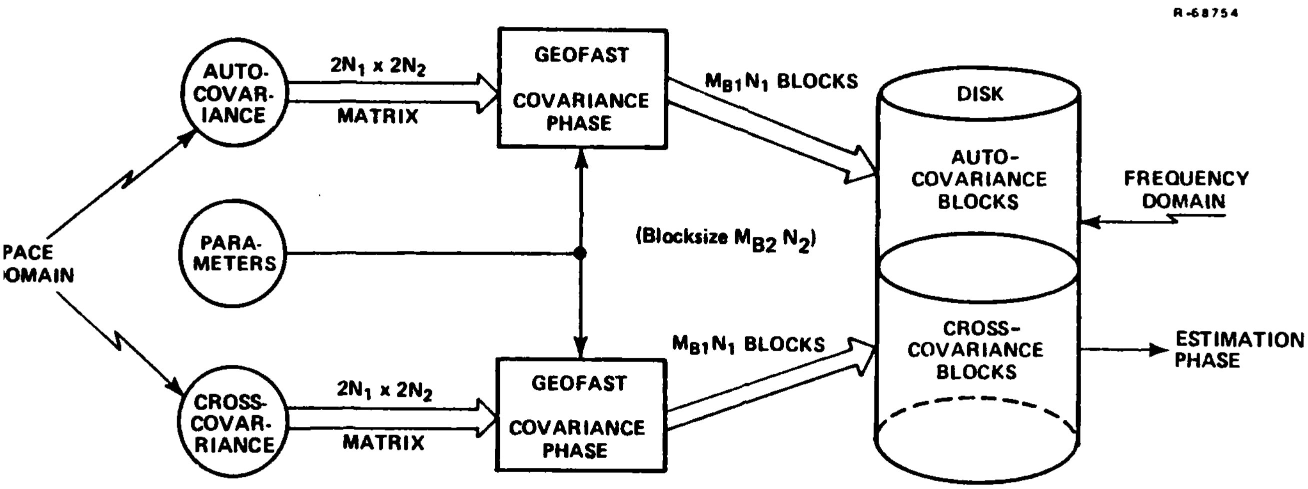


Figure 5.2-2 GEOFAST Covariance Phase

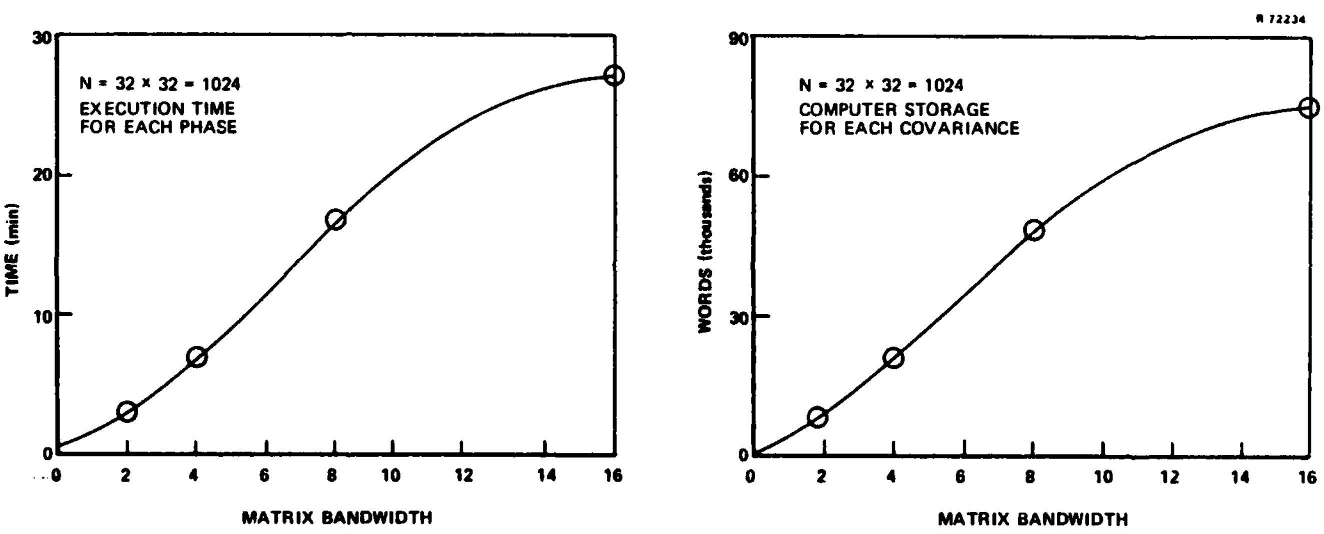


Figure 5.2-3 Time and Storage Variation With Bandwidth

stores the resulting matrix on disk for use in the estimation phase (GEOEST3). The program is run once for the autocovariance and once for the crosscovariance (see Fig. 5.2-2).

Parameter Input

The list of parameters required by GEOCOV3 is shown in Table 5.2-1. For each parameter the data type, dimension, default value, and restrictions are given. Mandatory inputs are an alphabetic identifier and the matrix dimensions. The present version of the code accepts row and column dimensions of 4, 8, 16, and 32 only. This may be increased to any power of two by recompiling the main program (see program documentation).

Increasing the matrix bandwidth parameter (MB) increases accuracy, execution time, and computer storage requirements. Typical time and storage variations with bandwidth is shown in Fig. 5.2-3. Accuracies are, in general, problem dependent, but full accuracy is attained when the bandwidth is one half the dimension.

Significant savings occur when the matrix is symmetric or antisymmetric and the indicator (ISWTCH) is properly set. For an autocovarance matrix the indicator must be set to symmetric (ISWTCH=2).

Matrix Input

The GEOFAST algorithm assumes that the covariance functions are shift-invariant and that the data are gridded, with the result that the covariance matrices may be specified by only 4N elements, where N is the number of grid locations. The format for the covariance matrix input will now be given.

```
CASE IDENTIFICATION HEADER
 CURRENT VALUE IS  TEST GEOFAST ESTIMATION
*****ENTER NEW VALUE OR <CR>*****

AUTO COVARIANCE FILE NAME
 CURRENT VALUE IS
*****ENTER NEW VALUE OR <CR>*****
AUTO
 THE NEW VALUE IS AUTO

CROSS COVARIANCE FILE NAME
 CURRENT VALUE IS
*****ENTER NEW VALUE OR <CR>*****
CROSS
 THE NEW VALUE IS CROSS

INPUT DATA FILE NAME
 CURRENT VALUE IS
*****ENTER NEW VALUE OR <CR>*****
DATA
 THE NEW VALUE IS DATA

OUTPUT ESTIMATES FILE NAME
 CURRENT VALUE IS
*****ENTER NEW VALUE OR <CR>*****
ESTIMATES
 THE NEW VALUE IS ESTIMATES

CASE ROW DIMENSION
 CURRENT VALUE IS           16
*****ENTER NEW VALUE OR <CR>*****

CASE COLUMN DIMENSION
 CURRENT VALUE IS           16
*****ENTER NEW VALUE OR <CR>*****

MATRIX ROW BAND WIDTH
 CURRENT VALUE IS            5
*****ENTER NEW VALUE OR <CR>*****

MATRIX COLUMN BAND WIDTH
 CURRENT VALUE IS            5
*****ENTER NEW VALUE OR <CR>*****

DATA STREAM FILE NAME
 CURRENT VALUE IS
*****ENTER NEW VALUE OR <CR>*****
TESTEST
 THE NEW VALUE IS TESTEST

DO YOU WISH TO SUBMIT THESE JOBS
 CURRENT VALUE IS  Y
*****ENTER NEW VALUE OR <CR>*****
```

Figure 5.2-1   Sample GEOFAST Run (Continued)

```
GEOFAST PHASE INDICATOR
  Covariance - CREATE COVARIANCE DATA STREAM
  Estimation - CREATE ESTIMATION DATA STREAM
  Quit       - TERMINATE DATA ENTRY
 CURRENT VALUE IS  COV
*****ENTER NEW VALUE OR <CR>*****

CASE IDENTIFICATION HEADER
 CURRENT VALUE IS  TEST GEOFAST COVARIANCE
*****ENTER NEW VALUE OR <CR>*****

INPUT COVARIANCE FILE NAME
 CURRENT VALUE IS
*****ENTER NEW VALUE OR <CR>*****
COVINPUT
 THE NEW VALUE IS COVINPUT

OUTPUT COVARIANCE FILE NAME
 CURRENT VALUE IS
*****ENTER NEW VALUE OR <CR>*****
 COVOUTPUT
  THE NEW VALUE IS COVOUTPUT

CASE ROW DIMENSION
 CURRENT VALUE IS          16
*****ENTER NEW VALUE OR <CR>*****

CASE COLUMN DIMENSION
 CURRENT VALUE IS          16
*****ENTER NEW VALUE OR <CR>*****

MATRIX ROW BAND WIDTH
 CURRENT VALUE IS           5
*****ENTER NEW VALUE OR <CR>*****

MATRIX COLUMN BAND WIDTH
 CURRENT VALUE IS           5
*****ENTER NEW VALUE OR <CR>*****

MATRIX SYMMETRY INDICATOR
  1 - NO SYMMETRY
  2 - SYMMETRIC
  3 - ANTI-SYMMETRIC
 CURRENT VALUE IS           2
*****ENTER NEW VALUE OR <CR>*****

DATA STREAM FILE NAME
 CURRENT VALUE IS
*****ENTER NEW VALUE OR <CR>*****
 TESTCOV
  THE NEW VALUE IS TESTCOV

GEOFAST PHASE INDICATOR
  Covariance - CREATE COVARIANCE DATA STREAM
  Estimation - CREATE ESTIMATION DATA STREAM
  Quit       - TERMINATE DATA ENTRY
 CURRENT VALUE IS  COV
*****ENTER NEW VALUE OR <CR>*****
 EST
 THE NEW VALUE IS  EST
```

Figure 5.2-1 Sample GEOFAST Run

job uniquely. The files so created may be modified later by the EDT command and resubmitted directly if desired.

The GEOFAST program is illustrated by the sample run shown in Fig. 5.2-1. Before each response the user is given a description and the current (or default) value and asked for a new value. A carriage return indicates acceptance of the current value. A complete list of parameters and default values is given in Tables 5.2-1 and 5.2-2 (sections 5.2.2 and 5.2.3). Any parameters not prompted for are automatically set to default values. The user may override these parameter settings by editing the data stream file created by GEOFAST prior to submission.

All file names default to the current directory and file type DAT. If these defaults are inappropriate the user must supply the full filename.

Errors and Diagnostic Messages

The GEOFAST program checks all parameter inputs for conformity to type and restrictions as indicated in Tables 5.2-1 and 5.2-2. When an error is detected, a diagnostic message is printed and the user is prompted to enter a new value. The diagnostic messages are sufficiently self-explanatory that they are not listed here.

5.2.2 The GEOCOV3 Program

Purpose

This program is the covariance phase of the GEOFAST estimation algorithm. It transforms an input covariance matrix from the space domain to the frequency domain and

- The covariance input files CZZ, CXZ are described under GEOCOV3
- The data input file Z is described under GEOEST3.

The descriptions include all basic information necessary to execute the programs, together with a discussion of errors and diagnostic output.

5.2.1 The GEOFAST Command Program

Purpose

This program is run interactively to create and submit the batch jobs that implement the GEOFAST estimation algorithm (GEOCOV3 and GEOEST3). The program prompts for all required inputs, supplies defaults as requested, and checks for possible errors and inconsistencies.

Program Execution

The GEOFAST program is executed by the RUN command and is then controlled by user responses to prompt messages. The program allows the user to create and submit up to two covariance phase jobs and one estimation phase job in a single execution. The covariance jobs will be run before the estimation job. Alternatively, each phase may be submitted separately.

For each job submitted, a data stream file is created which contains the names of any matrix input or output files required (that is CZZ, CXZ, Z, and X) together with any parameters required. These entries are constructed from the user responses and the default values. In addition, the user is requested to supply a data stream filename to identify each

5.2 GEOFAST PROGRAMS

The GEOFAST software package consists of four separate programs.

- GEOFAST is an interactive user-oriented program which creates and submits the batch jobs to perform GEOFAST estimation. (See section 5.2.1)
- GEOCOV3 is the covariance phase of the GEOFAST algorithm (See section 5.2.2)
- GEOEST3 is the estimation phase of the GEOFAST algorithm (See section 5.2.3)
- GEOPLOT is an interactive user-oriented program which produces contour and surface plots of data files. (See section 5.2.4)

The GEOFAST program is all that will be needed by most users. This interactive program prompts for all required inputs and supplies defaults as requested. User inputs are checked for possible errors and used to create data streams and submit jobs for the two phases of processing (GEOCOV3 and GEOEST3).

The covariance phase (GEOCOV3) and the estimation phase (GEOEST3) may also be executed independently, either in interactive (RUN) mode or batch (SUBMIT) mode. To do so the user must either create a data stream or modify an existing data stream. For the user familiar with the GEOFAST system this may be a more efficient mode of execution.

The next sections describe the four programs listed above. The preparation of input files is covered under the appropriate processing phase, namely

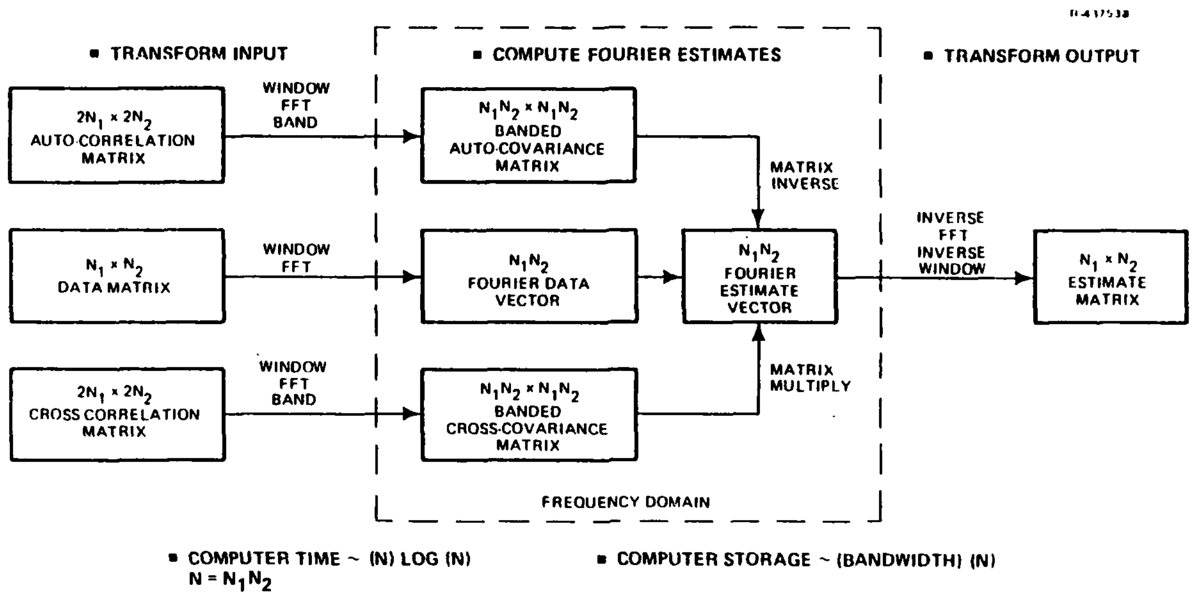


Figure 5.1-1   Overview of the GEOFAST Algorithm

which are described more fully below. These inputs are combined via the minimum-variance estimation equations (least-squares collocation) to produce the output matrix, X, a gridded estimate matrix, which corresponds to the same grid locations as Z (possibly shifted in origin and altitude).

In order to run the GEOFAST software the user must create input files containing the three input matrices in formats described below, and designate an output file to receive the output matrix. In addition, parameters controlling the processing must be supplied, or the user may elect to take automatically supplied default options. The next section gives an overview of the software package, together with instructions for input preparation and program execution.

5. USER'S GUIDE TO THE GEOFAST ESTIMATION SOFTWARE

5.1 OVERVIEW OF GEOFAST CAPABILITIES

The GEOFAST algorithm provides the capability for rapid calculation of minimum-variance estimates from high-density gravimetric data. The GEOFAST method uses frequency domain techniques to achieve a complexity of order N log N where N is the number of data points. This efficiency is obtained by exploiting two structures commonly found in geodetic applications:

- Input data that are regularly spaced on a Cartesian grid (after preprocessing)
- Statistical covariance models that are stationary (or shift-invariant)

The resulting algorithm is implemented in two dimensions and is well suited to the processing of large gravity data bases. A balance between accuracy and computer time may be controlled by the choice of design parameters. A complete description of the algorithm, including the mathematical background and selected test cases using real data, is contained in Ref. 8.

An overview of the GEOFAST algorithm is shown schematically in Fig. 5.1-1. There are three input matrices

- A gridded data matrix, Z
- An autocovariance matrix, CZZ
- A crosscovariance matrix, CXZ

Matrix Input

The GEOFAST algorithm assumes that the measurement data are gridded. The data can therefore be specified as a matrix Z(I,J) where I and J are row and column subscripts of dimension N1 and N2 respectively. The elements of the matrix Z must be stored as double precision (REAL*8) numbers in column order in an unformatted file. If the matrix Z is created by a FORTRAN program as a double precision array and written to disk by a statement of the form "WRITE (IUNIT) Z' then the resulting file is in the correct form to be read by GEOEST3 (see decription of subroutine ZINPUT in the proram documentation for further details). The input data matrix is also normally printed.

Program Execution

The GEOEST3 program may be executed interactively by the RUN command. In this mode the user is prompted for a data stream file containing the parameter input values. Included in this file are the names of the matrix input files and the name of the matrix output file. The required formats can be obtained by creating the data stream file through the GEOFAST command program.

The GEOEST3 program may also be executed in the batch mode. In this mode the user must create and submit a command file of the form

```
$SET DEFAULT [DIRECTORY]
$RUN GEOEST3
FILE-NAME
```

where DIRECTORY is the name of the current file directory and FILE-NAME is the name of the data stream file containing the

parameter values and matrix filenames. All files should reside in the current directory or be explicitly named.

Matrix Output

The estimates X(I,J) produced by GEOEST3 are written to an output file designated by the user in the same format as the Z matrix input, and normally also printed. The current version of GEOEST3 sets the first window element to zero (see description of subroutine WINDOW in the program documentation for further details). This results in zeroing the first row and column of both Z and X. These zeros shoul be deleted when plotting or processing these matrices. (See GEOPLOT program).

Errors and Diagnostic Messages

Parameter input is checked for errors in the GEOFAST command program, but no during execution of GEOEST3. However, consistency is checked between dimensions and banwidths as specified in the parameter input and as recorded in a header record associated with each covariance file. An error message results if the parameters are inconsistent or if the autocovariance matrix was not declared symmetric (see GEOCOV3 parameter input).

If the matrix inversion is terminated because of ill-conditioning, a message

ILL-CONDITIONED MATRIX:
(CO)SINE INVERSE TRUNCATED AT ROW (COLUMN) = K

is provided, indicating pivot failure in one of the covariance submatrices at row (or column) K. If a frequency limit was specified at row (or column) K, only the second line of this message is printed.

The value of the covergence criterion (which is compared to CEPS) and the number of iteration steps (limited by MSTEPS) are also printed in the form:

ITERATION COMPLETE: NUMBER OF STEPS = M
RELATIVE RMS RESIDUAL = R

Increasingly detailed printout of intermediate calculations may be obtained by setting PRTFLG = 1, 2, or 3. Setting PRTFLG = -1 will suppress printing of the input data matrix and the output estimates matrix.

5.2.4 The GEOPLOT Command Program

Purpose

All matrix input and output files (CZZ, CXZ, Z, and X) associated with the GEOFAST program can be plotted with the WSS graphics plotting package. The purpose of this routine is to provide rapid and simple plotting capability for the input data matrix Z and output estimates matrix X.

Program Execution

The GEOPLOT program is executed interactively by the RUN command. The user is prompted for the name of the file to be plotted and its matrix dimensions. Two plots are generated:

- A contour plot with automatic contour level selecticn
- A surface plot viewed from an elevation of 45 deg and an azimuth of -45 deg.

Before plotting, the first row and column of the matrix are deleted. The user is prompted for a plot title which defaults to the given filename.

The user is also prompted for horizontal axis labels and scales. These default to LATITUDE and LONGITUDE with scales from 1 to N. Scaling of contour levels and surface height is automatic, based on the data values. The user is prompted for a vertical axis label which defaults to VALUE.

The program first displays each plot on the graphics screen and then provides the user with the standard plot package menu:

<CR>/CLEAR	-	to clear screen and continue
PRINT	-	to produce hard copy
ZOOM	-	to zoom and scroll
SCALE	-	to change axis scaling
BACK	-	to return to command input
QUIT	-	to terminate the program

The commands used in GEOPLOT are:

<CR>/CLEAR - This will clear the Lexidata screen. If there is another plot in the series it will be presented, otherwise control will go back to the command input line with the "Enter command: " prompt.

PRINT - A hard copy of the current plot shown on the Lexidata will be printed on the cline printer after the program stops.

ZOOM - The user has the ability to expand any area of the plot. After entering ZOOM, the user sets any WHITE switch on the trackball unit to the ON position. The trackball can now be used to position any particular part of the plot on the screen. If a higher magnification is subsequently set, the area

expanded will be the upper right corner of the screen. The WHITE switches set the magnification level, with switch E (the first WHITE switch to the left of the BLUE one) having a unity magnification and magnification increasing by two with switch A (rightmost WHITE switch) having a magnification factor of 16. When the user is finished with the zoom facility, all switches should be set to their OFF position and the BLUE switch should be toggled to redisplay the plot menu on the VT100.

QUIT - This command will terminate the program execution.

The user may zoom and scroll as desired, as explained above. Using the blue switch on the trackball unit causes the program to redisplay the plot menu. Typing PRINT (or P) will print a hard copy of the original plot (without zoom and scroll). The user enters a carriage return to proceed to the next plot. After the second plot, the user is given an option to generate a surface plot from a different viewing angle. There is no error messages other than those contained in the plotting software package itself.

APPENDIX A
LIST OF MULTISENS INPUT PARAMETERS

NAME	TYPE	DIM	DEFAULT	DESCRIPTION
General run options				
TYPE_OF_RUN	Char	3	None	Phases of program to be executed
TRAJFILE	Char	-	None	Input trajectory filename
TRANSFILE	Char	-	Note 1	Inflight tranfer function filename
GRAVFILE	Char	-	Note 2	Geoid undulation density
USERFILE	Char	3	Note 3	Input filename(s) for NUMERIC model
TRANSFER phase parameters				
TDEPLOY	Real	-	400.	Deployment time in sec from launch
LAMBDAO	Real	-	2400.	Longest non-DC wavelength (nm)
CRIT	Real	-	1.E-2	Criterion for attenuation factor
NFM1	Integer	-	10	Control for number of table entries
NWAVE	Integer	-	12	Subdivisions of wavelength
DEBUG	Logical	-	.FALSE.	TRANSFER debug print switch
MDEBUG	Integer	-	0	TRANSFER debug print loop index
General GRAVITY phase parameters				
GRAVERR	Char	10	UND	List of gravimetric quantities
MODE	Char	-	RMS	Gravity statistics run option
TRKANGL	Real	-	0.	Track angle in deg from east
ALTITUDE	Real	-	0.	Altitude for gravimetric errors (m)
EL	Real	-	0.	Mean latitude of estimation region (deg)
GRIDSAVE	Real	-	1.	Integration grid change factor
JPRINT	Integer	-	0	GRAVITY debug print level
GRAVITY integration region definition (For MODE=PSD or COV only)				
M1	Integer	-	100	Number of frequency increments alongtrack
M2	Integer	-	100	Number of frequency increments crosstrack
SAMPLEA	Real	-	1.	Half mininimum wavelength alongtrack (nm)
SAMPLEC	Real	-	1.	Half mininimum wavelength crosstrack (nm)
Area mean option parameter				
AREASIZE	Real	-	Note 4	Side of area mean bin ($\widehat{min}$).
Gravity model parameters				
MODEL	Char	-	BASELINE	Gravity model descriptor
SIGMAN12	Real	-	Note 5	Variance 1st Markov undulation model

SIGMAN22	Real	-	Note 5	Variance 2nd Markov undulation model
BETAN1	Real	-	Note 5	Reciprocal characteristic distance 1st Markov undulation model (1/m)
BETAN2	Real	-	Note 5	Reciprocal characteristic distance 2nd Markov undulation model (1/m)
SIGMAC2	Real	-	0.36	Variance Markov ocean current model (m^2)
BETAC	Real	-	2.3809524E-5	Reciprocal characteristic distance ocean current model (1/m)
USDFACT	Real	-	1.	Scale factor for NUMERIC model

Plot option parameters

PLOT	Char	7	None	Plot descriptors
PLOTFILE	Char	-	None	Filename for output plot file
M1PLOT	Integer	-	50	Number of east spacings for 3-D, CONTOUR
M2PLOT	Integer	-	50	Number of north spacings for 3-D, CONTOUR

Survey options

SURVEY	Char	5	None	List of survey descriptors

Parameters for GEOSALT survey

ORBINC	Real	-	115	Satellite orbit inclination (deg)
HEITALT	Real	-	8.E5	Satellite altitude (m)
TRKSP	Real	-	30	Separation of equator crossings (nm)
SAMPALT	Real	-	0.1	Time between samples (sec)
CA	Real	-	0.36	Altimeter white-noise level (m^2)
ALTVAR	Real	-	0.25	Variance of altimeter bias (m^2)
ALAM	Real	-	1.E-6	Inverse standard deviation bias model (1/m)

Parameters for SEASATALT survey

ORBINCP	Real	-	108	Satellite orbit inclination (deg)
HEITALTP	Real	-	8.E5	Satellite altitude (m)
TRKSPP	Real	-	80	Separation of equator crossings (nm)
SAMPALTP	Real	-	0.1	Time between samples (sec)
CAP	Real	-	0.01	Altimeter white noise level (m^2)
ALTVARP	Real	-	0.25	Variance of altimeter bias (m^2)
ALAM	Real	-	1.E-6	Inverse standard deviation bias model (1/m)

Parameters for GRAVSHIP survey

SE	Real	-	0.25	East data spacing (nm)
SN	Real	-	6	North data spacing (nm)

CG	Real	-	1.E-12	White noise level $([m/sec^2]^2)$
QQ	Real	-	0.75	RMS Eotvos error (mgal/measurement)
GRAVXTENT	Real	-	150	Maximum wavelength in model (nm)
EOTVOSW	Logical	-	.TRUE.	Eotvos error term switch
FULFUNC	Integer	-	0	Aliasing term switch

Parameters for GRAVLAND survey

SE	Real	-	81	East data spacing (nm)
SN	Real	-	81	North data spacing (nm)
CG	Real	-	9.E-10	White noise level $([m/sec^2]^2)$
GRAVXTENT	Real	-	-10	Maximum wavelength (no limit if negative)
FULFUNC	Integer	-	1	Aliasing term switch

Parameters for SST survey

LOORBINC	Real	-	86	Low satellite orbit inclination (deg)
LOHEIGHT	Real	-	150	Low satellite altitude (km)
SSTTRKSP	Real	-	30	Separation of equator crossings (nm)
SSTSP	Real	-	10	Time between measurements (sec)
HIHEIGHT	Real	-	3.5786E7	High satellite altitude (m)
SSTLONG	Real	-	0	Region longitude relative to high satellite (deg)
SSTNOISE	Real	-	100	RMS white-noise level (micrometer/sec)
SSTFUNC	Integer	-	0	Switch to activate full aliasing

Parameters for Gradiometer surveys

GRADSP	Real	-	10	Distance between tracks (km)
SAMPINT	Real	-	10	Time between samples (sec)
SPEED	Real	-	555.6	Speed of aircraft (km/hr)
HEIGHT	Real	-	6096	Altitude of aircraft (m)
GRADXTENT	Real	-	1500	Maximum wavelength (nm) (No limit if negative)
GRADFUNC	Integer	-	1	Switch for full aliasing calculation
SWGRAD	Logical	3	Note 6	Switches for gradiometer components
GRADROT	Logical	-	Note 7	Switch for umbrella geometry
RDNSE	Real	6	Note 8	Red noise values for GRAD survey
WHTNSE	Real	6	Note 8	White noise values for GRAD survey

Parameter for IMPACT phase

WAVELONG	Real	-	2700	Maximum wavelength to be included (nm)

Parameters for Sensitivity option

SENSITIVITY	Char	-	None	Descriptor for dependent variable
VARIABLE	Char	-	None	Name of independent variable
VALUE	Real	20	None	List of values for independent variable

Notes:

1) For a run including the TRANSFER phase, TRANSFILE defaults to a temporary file. For a run including the IMPACT phase but not the TRANSFER phase, a filename must be supplied for TRANSFILE.

2) For a run including the GRAVITY phase, GRAVFILE defaults to a temporary file; if MODE=RMS or AREAMEAN, this default cannot be overridden. For a run including the IMPACT phase but not the GRAVITY phase, a filename must be supplied.

3) At least one USERFILE filename must be supplied for a MODEL=NUMERIC run; if MODE=RMS or AREAMEAN, up to three such files may be used, one for each of the regions of integration (see text).

4) For a sensitivity run, AREASIZE defaults to 60 $\widehat{\text{min}}$. For all other runs, if AREASIZE is not specified in the input, gravimetric covariances will be computed using AREASIZE values of 5, 15, 60, and 300 $\widehat{\text{min}}$. If AREASIZE is specified, only the single value entered will be used.

5) Default values for SIGMN12, SIGMN22, BETAN1, and BETAN2 depend on whether the BASELINE or the ACTIVE model is selected; they are listed in the text. These parameters are not used for other gravity models.

6) If not specified, all SWGRAD switches default to TRUE. However if, some of them are set to TRUE by the input, the others default to FALSE.

7) For Draper surveys, GRADROT is set to TRUE and cannot be overridden; for the other gradiometer surveys, it defaults to FALSE but may be overridden.

8) For all gradiometer types except GRAD, the parameters RDNSE and WHTNSE are set by the program and cannot be overridden. For the user-defined survey GRAD, values must be supplied for these parameters.

APPENDIX B
DESCRIPTION OF NAMELIST FACILITY

B.1 NAMELIST GENERAL INFORMATION

For batch input, a pseudo-NAMELIST facility is available, which combines some of the features of IBM's FORTRAN NAMELIST and the PL/I GET DATA. It provides the following features.

- The input may be free-form and is independent of card boundaries. (All 80 columns of each input recorded are scanned, except that the scan is terminated when an exclamation point is encountered, in order to permit comment fields in the input.)

- The variables whose values are to be set are identified by their names, and so may be entered in any order. Any subset of the listed scalar variables or array elements may be specified by the user; the values of those not specified are not altered by the input operation. The input stream is terminated by a semicolon.

- Each input record is echoed on the print file, and errors in format, syntax, spelling, etc., are flagged immediately following the card in which they were detected. Processing of input continues following an error, but at the end of the operation an error flag is returned to the calling program, which may then terminate the execution.

- Variable types which may be set are: INTEGER*2, INTEGER*4, REAL*4, REAL*8, COMPLEX*8, COMPLEX*16, LOGICAL*1, LOGICAL*4, and CHARACTER*n for $n \leq 80$.

- Subscript checking is performed on array elements entered, so that array boundaries are not violated.

- Comment fields may be included in the input stream. All characters on a line to the right of an exclamation point are treated as comments, and are printed but not scanned. For this reason, exclamation points should not be embedded within the data to be scanned.

- A corresponding pseudo-NAMELIST print facility is also provided, which generates a labeled printout of the contents of all variables and arrays in the list.

B.2 FORMAT FOR BATCH INPUT

Input to the operation consists of a series of equations of the form: name = value. These equations may be separated by blanks and/or commas, and the series is terminated by a semicolon. Since card boundaries are ignored, any part of an equation may be continued from column 80 of one card to column 1 of the next noncomment card; hence card boundaries are not in themselves sufficient to separate equations. Blanks embedded within the name field, or around the equals sign, are ignored; blanks embedded within the value field may be used to separate elements in a list of values, or may be part of a character-string value, as described below.

The name field consists of one or more variable names (any of which may be subscripted) separated by commas. The variable names themselves may consist of any printable characters other than commas, parentheses, or semicolons. If more than one name is present, the values indicated in the value field are assigned to each variable indicated in the name field. For example, the equation:

$$A(2),A(5),B,C = 1$$

ould assign the value 1 to the second and fifth elements of
ne array A, and to the variables B and C. If B or C was an
rray, the value 1 would be assigned to only the first elements
f these arrays. The current implementation is restricted to
ingle subscripts; however, values may be assigned to elements
f multidimensional arrays by providing an equivalent single
ubscript. For example, if the array A were dimensioned (2,2,3),
ne above equation would assign the value 1 to elements (2,1,1)
nd (1,1,2) of that array. Note that this implementation also
ssumes that the arrays are dimensioned beginning with the
ndex 1 -- no allowance is made for zero or negative indices.

The value field is a list of one or more values to be
ssigned to the indicated variables. If more than one value
s provided, they may be separated by commas and/or blanks,
nd the entire list must be enclosed in parentheses. Each value
ay be preceded by a repetition factor and an asterisk; in
uch a case no blanks are permitted around the asterisk. If
ultiple values are listed (or implied by a repetition factor)
he values will be assigned to consecutive elements of the
rray(s) indicated in the name field. If the array name is
ubscripted, the values will be assigned to consecutive array
lements beginning with the element indicated.

.3 EXAMPLES OF BATCH INPUT

ssume the following list of variables:

```
REAL*4 A(10),B(5,5)
CHARACTER*12 INPUTS(5),OUTPUTS(5)
LOGICAL*1 DEBUG,PLOTS,WRITEFILE
COMPLEX*8 CVAR,COVAR(5,5)
```

```
INTEGER*4 NTIMES,MAXLENGTH,INDEX(5)
```

the following inputs are all valid:

- The equations A=(5,5,5),A=3*5, and A(1)=3*5 are equivalent. Each sets the first three elements of A to the value 5.

- The equation A(3)=(2*4,3*2) will set elements 3 and 4 to the value 4, and elements 5, 6, and 7 to the value 2.

- The equation A(9)=(3,2*6) will set element 9 to the value 3, element 10 to the value 6, and generate an error message to indicate that assigning the third value would violate the array boundary.

- If B is a 5 by 5 matrix, the equation:

  B = (1, 5*0, 1, 5*0, 1, 5*0, 1, 5*0, 1)

  will set B to the identity matrix.

- ```
  INPUTS=(UND,NORTHVD,EASTVD)
  OUTPUTS=(POWER,CROSS-SPECT)
  ```

- ```
  DEBUG,PLOTS=NO,WRITEFILE=YES
  ```

- ```
  CVAR=1.2-0.5I
  COVAR=(5          2.1-0.4I 3*0
         2.1+0.4I 3         3*0
         2*0                1.       2-4I     I
         2*0                2+4I     6        5+0.02I
         2*0                -I       -5-0.2I  8.E0)
  ```

- ```
  NTIMES=100     MAXLENGTH=20    INDEX=(-1,2*1,0,2)
  ```

SYNTAX FOR VALUES

- Repetition factors are required to be positive integers. They are decoded in I format, and therefore may not contain decimal points. There should be no embedded blanks around the asterisk

separating the repetition factor from the value to which it applies.

- Values to be assigned to integer variables are decoded in F format, and so may contain decimal points or exponents, provided that the value does not exceed the largest magnitude that an integer variable may hold. If a noninteger value is entered, the value is truncated and a warning message is printed.

- Values to be assigned to real variables are decoded in F format. They may optionally contain decimal points or exponents.

- Values to be assigned to complex variables are entered in the form a+bI (or a+bJ) with no embedded blanks. The substrings a and b may be omitted, or may be real values, as illustrated in the following examples:

3	is equivalent to 3+0I
3+I	is equivalent to 3+1I
-I	is equivalent to 0-1I

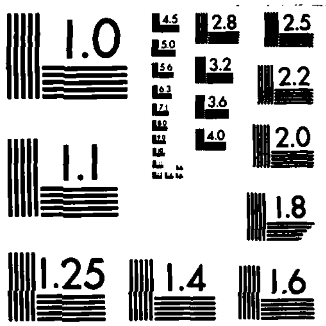
- Values to be assigned to logical variables may be indicated in a variety of ways. The value .TRUE. may be represented as .TRUE., TRUE, .T., T, YES, or Y. Similarly, the value .FALSE. may be represented as .FALSE., FALSE, .F., F, NO, or N. Any other specification generates an error message.

- Values to be assigned to character variables must be enclosed in single quotes if they contain blanks, commas, or asterisks· otherwise, the quotes may be omitted. Character values may not contain semicolons or exclamation points; if a quote is to be included as part of a character-string value, it should be represented as two consecutive quotes (e.g. 'DON''T DO IT'). If a value is assigned which exceeds the length of the character variable, truncation occurs but no error message is generated.

AD-A156 003 THE WEAPONS SUPPORT SYSTEM USER'S GUIDE(U) ANALYTIC SCIENCES CORP READING MA A R LESCHACK ET AL. 01 JUL 81 TASC-TR-1946-1 AFGL-TR-81-0225 F19628-80-C-0078 3/3

UNCLASSIFIED F/G 8/5 NL

END
FILMED
DTIC



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

- If a null value is coded, the value of the corresponding variable or array element is not altered, but retains its default value. For example, the stream: A=,B=(5,,,1); would leave the variable A and elements 2 and 3 of the array B unaltered, while setting elements 1 and 4 of B.

- If more than one variable is listed in the 'name' field, the variables need not be of the same type, provided that the value specifications after the equals sign are compatible with all the variables listed. For example, if R is a real variable, I an integer variable, L a logical variable, and C a character variable, the equations R,I,C=5 and L,C=TRUE are both valid.

B.5 RESTRICTIONS ON BATCH INPUT

The following limitations apply to the current implementation:

- Variable names are limited to 31 characters or less (excluding subscripts), corresponding to the limit of the FORTRAN compiler.

- The maximum number of characters in the name field is 80.

- The maximum number of names in the name field is 10.

- The maximum number of characters in the value field is 400.

- The longest character variable which can be assigned is CHARACTER*80.

REFERENCES

1. Goldstein, J.D., "Analysis and Simulation of Multisensor Gravity Surveys and Associated Ballistic Missile Impact Errors (U)," The Analytic Sciences Corporation, Report No. TR-868-1-3, December 1980.

2. Gelb, A. (Editor), Applied Optimal Estimation, MIT Press, Cambridge, 1974.

3. Heller, W.G., and Jordan, S.K., "Attenuated White Noise Statistical Gravity Model," Journal of Geophysical Research, Vol. 84, No. B9, August 1979.

4. White, J.V., "Error Models for Gravity Gradiometers in Airborne Surveys," Technical Report AFGL-TR-80-0220, Air Force Geophysics Laboratory, Hanscom AFB, January 1980. ADA097745

5. Brammmer, R.F., LeSchack, A.R., and Ofenstein, W.T., "Gravity Data Evaluation Analysis and the Phase-Zero Definition of the Physical Geodesy System," The Analytic Sciences Corporation, Report No. TR-1104-1, April 1978.

6. Brammmer, R.F., LeSchack, A.R., and Ofenstein, W.T., "Gravity Data Evaluation Analysis and the Software Design Plan for the Physical Geodesy System," The Analytic Sciences Corporation, Report No. TR-1240-1, April 1978.

7. TASC Graphics Software Package Reference Manual, a part of the WSS Documentation.

8. Tait, K.S., "A Fast Estimation Algorithm for Two-Dimensional Gravity Data (GEOFAST)", Technical Report AFGL-TR-80-0016, Air Force Geophysics Laboratory, 15 November 1979. ADA086835

END

FILMED

8-85

DTIC